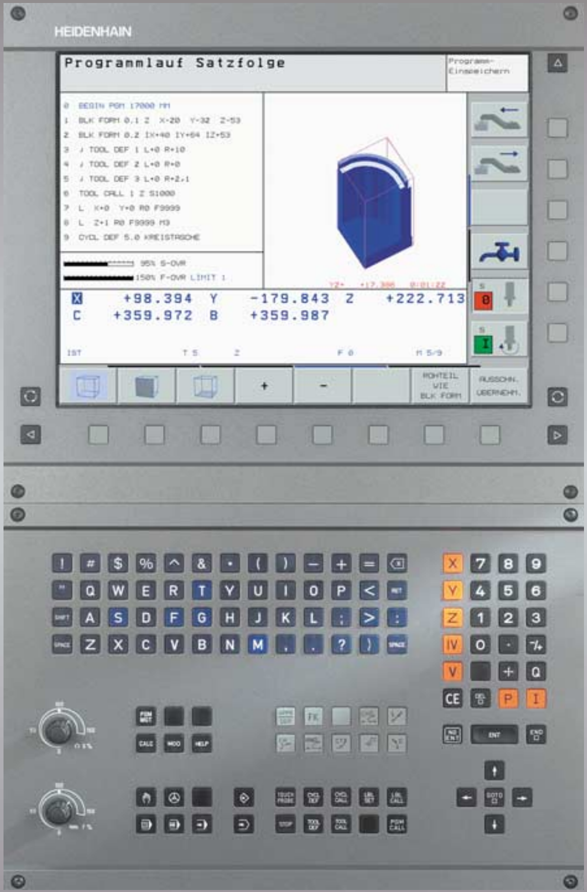
HEIDENHAIN



Pilot

iTNC 530

NC-Software
340 420-xx

English (en)
7/2002

The Pilot

... is your concise programming guide for the HEIDENHAIN iTNC 530 contouring control. For more comprehensive information on programming and operating, refer to the TNC User's Manual. There you will find complete information on:

- Q-parameter programming
- The central tool file
- 3-D tool compensation
- Tool measurement

Certain symbols are used in the Pilot to denote specific types of information:

	Important note
	WARNING: danger for the user or the machine!
	The TNC and the machine tool must be prepared by the machine tool builder to perform these functions!
	Chapter in User's Manual where you will find more detailed information on the current topic.

The information in this Pilot applies to TNCs with the following software numbers:

Control	NC Software Number
iTNC 530	340 420-xx

*) Export version

Contents

Fundamentals

Programs/Files

See "Programming, File Management"

The TNC keeps its programs, tables and texts in files.
A file designation consists of two components:

THREAD2.H

File name	File type
Maximum length: 16 characters	See table at right

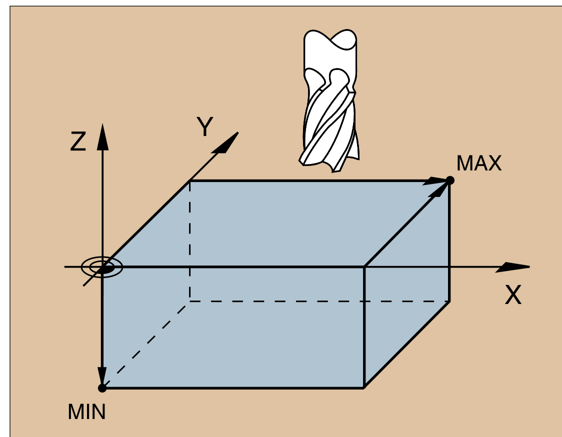
Files in the TNC	File type
Programs	
• in HEIDENHAIN format	**.H**
• in ISO format	**.I**
Tables for	
• Tools	**.T**
• Datums	**.D**
• Pallets	**.P**
• Cutting data	**.CDT**
• Positions	**.PNT**
Texts as	
• ASCII files	**.A**

Creating a New Part Program

PGM MGT

- Select the directory in which the program is stored
- Enter a new file name with file type
- Select unit of measure for dimensions (mm or inches)
- Define the blank form (BLK) for graphics:
  - Enter the spindle axis
  - Enter coordinates of the MIN point: the smallest X, Y and Z coordinates
  - Enter coordinates of the MAX point: the greatest X, Y and Z coordinates

```
1 BLK FORM 0.1 Z X+0 Y+0 Z-50
2 BLK FORM 0.2 X+100 Y+100 Z+0
```



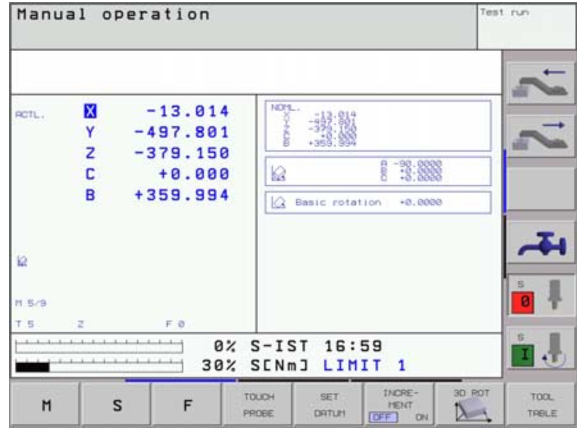
Choosing the Screen Layout

See "Introduction, the iTNC 530"

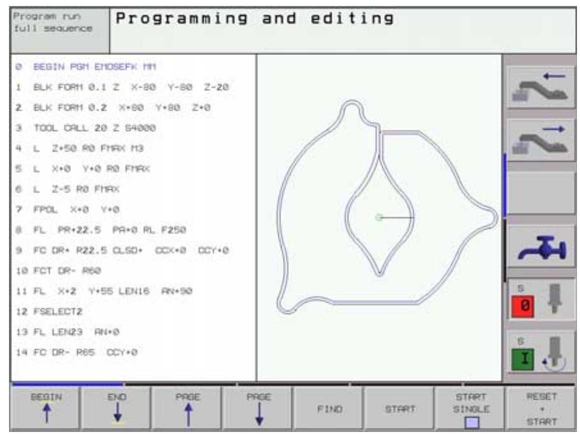
▶ Show soft keys for setting the screen layout

Mode of operation	Screen contents	
Manual operation Electronic handwheel	Positions	POSITION
	Positions at left Status at right	POSITION + STATUS
Positioning with manual data input	Program	PGM
	Program at left Status at right	PGM + STATUS
Program run, full sequence	Program	PGM
Program run, single block test run	Program at left Program structure at right	PGM + SECTS
	Program at left Status at right	PGM + STATUS
	Program at left Graphics at right	PGM + GRAPHICS
	Graphics	GRAPHICS

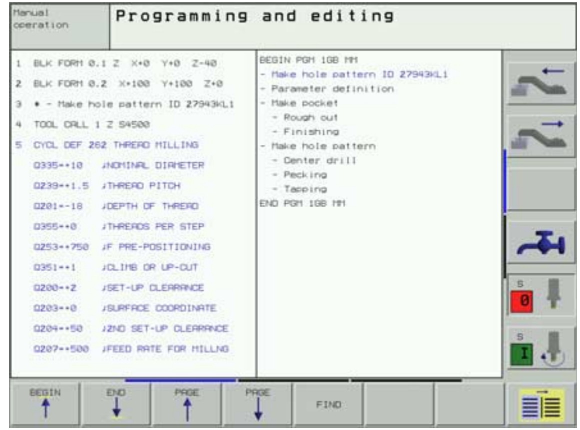
Continued ▶



▲ Positions at left, status at right
▼ Program at left, graphics at right



Mode of operation	Screen contents	
Programming and editing	Program	PGM
	Program at left Program structure at right	PGM + SECTS
	Program at left Programming graphics at right	PGM + GRAPHICS



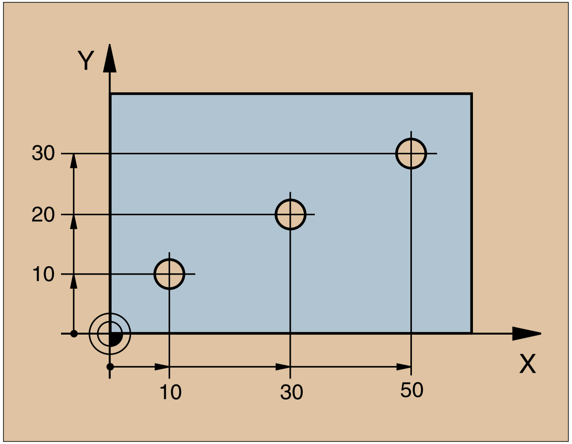
▲ Program at left, program structure at right

Absolute Cartesian Coordinates

The dimensions are measured from the current datum.
The tool moves **to** the absolute coordinates.

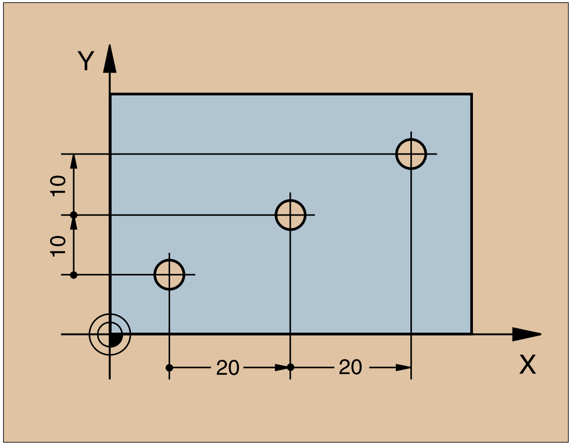
Programmable axes in an NC block

Linear motion: 5 axes
Circular motion: 2 linear axes in a plane or
3 linear axes with cycle 19 WORKING PLANE



Incremental Cartesian Coordinates

The dimensions are measured from the last programmed position of the tool.
The tool moves **by** the incremental coordinates.



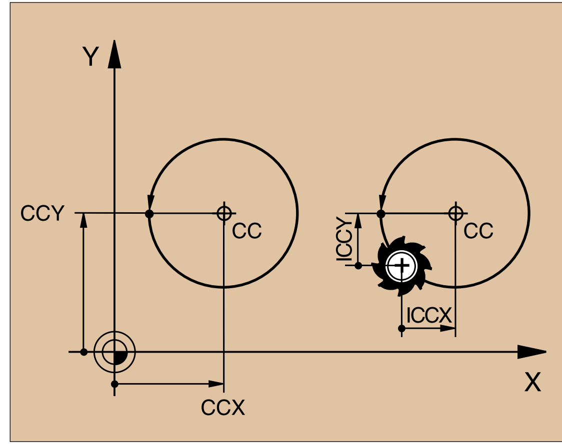
Circle Center and Pole: CC

The circle center (CC) must be entered to program circular tool movements with the path function C (see page 21). CC is also needed to define the pole for polar coordinates.

CC is entered in Cartesian coordinates*.

An absolutely defined circle center or pole is always measured from the workpiece datum.

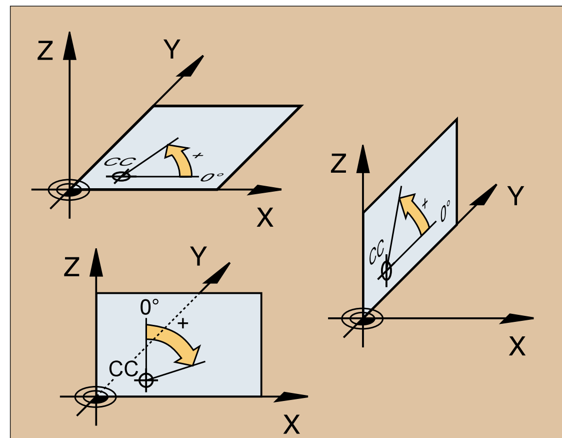
An incrementally defined circle center or pole is always measured from the last programmed position of the workpiece.



Angle Reference Axis

Angles—such as a polar coordinate angle PA or an angle of rotation ROT—are measured from the angle reference axis.

Working plane	Ref. axis and 0° direction
X/Y	X
Y/Z	Y
Z/X	Z

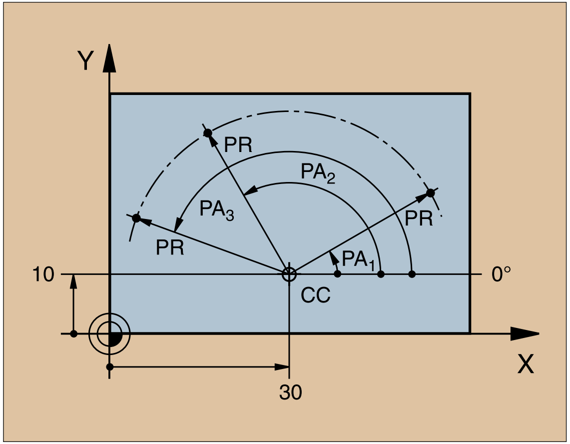


*Circle center in polar coordinates: See FK programming

Polar Coordinates

Dimensions in polar coordinates are referenced to the pole (CC).
A position in the working plane is defined by

- Polar coordinate radius PR = Distance of the position from the pole
- Polar coordinate angle PA = Angle from the angle reference axis to the straight line CC – PR



Incremental dimensions

Incremental dimensions in polar coordinates are measured from the last programmed position.

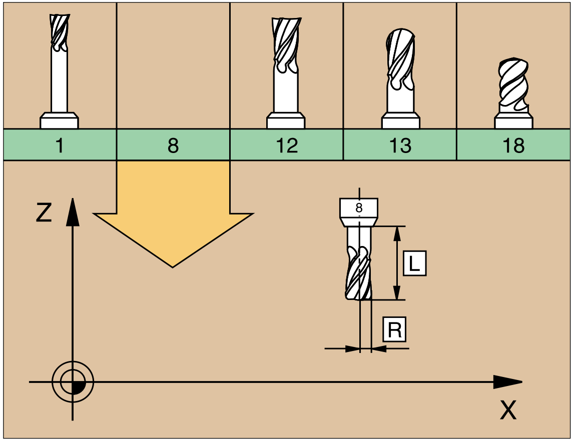
Programming polar coordinates

▶ Select the path function

▶ Press the P key

▶ Answer the dialog prompts

Defining Tools



Tool data

Every tool is designated by a tool number between 1 and 254 or, if you are using tool tables, by a tool name.

Entering tool data

You can enter the tool data (length L and radius R)

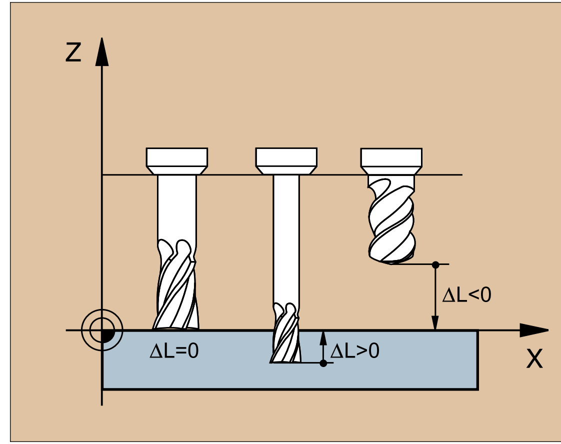
- in a tool table (centrally, Program TOOL.T)
  or
- within the part program in TOOL DEF blocks (locally)

TOOL DEF

- Tool number
- Tool length L
- Tool radius R

- Program the tool length as its difference ΔL to the zero tool:
  ΔL>0: The tool is longer than the zero tool
  ΔL<0: The tool is shorter than the zero tool

- With a tool presetter you can measure the actual tool length, then program that length.



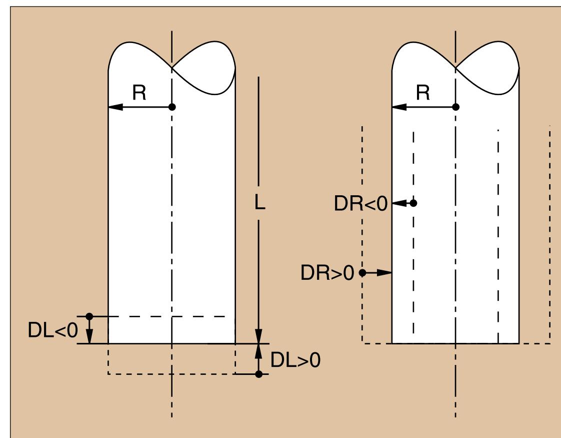
Calling the tool data

TOOL CALL

- Tool number or name
- Working spindle axis: tool axis
- Spindle speed S
- Feed rate
- Tool length oversize DL (e.g. to compensate wear)
- Tool radius oversize DR (e.g. to compensate wear)

```
3 TOOL DEF 6 L+7.5 R+3
4 TOOL CALL 6 Z S2000 F650 DL+1 DR+0.5
5 L Z+100 R0 FMAX
6 L X-10 Y-10 R0 FMAX M6
```

▼ Oversizes on an end mill



Tool change

- Beware of tool collision when moving to the tool change position!
- The direction of spindle rotation is defined by M function:
  M3: Clockwise
  M4: Counterclockwise
- The maximum permissible oversize for tool radius or length is ± 99.999 mm!

Tool Compensation

The TNC compensates the length L and radius R of the tool during machining.

Length compensation
Beginning of effect:

▶ Tool movement in the spindle axis

End of effect:

▶ Tool exchange or tool with the length L=0

Radius compensation
Beginning of effect:

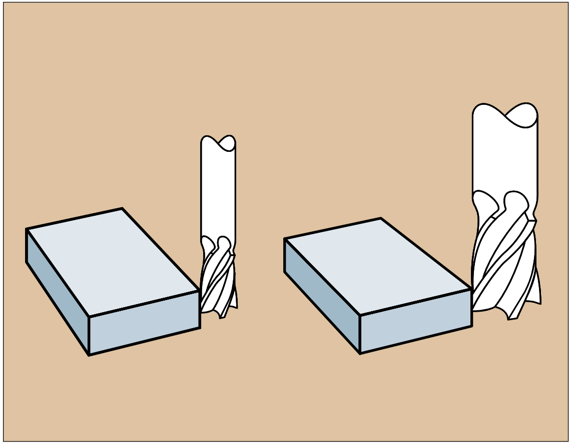
▶ Tool movement in the working plane with RR or RL

End of effect:

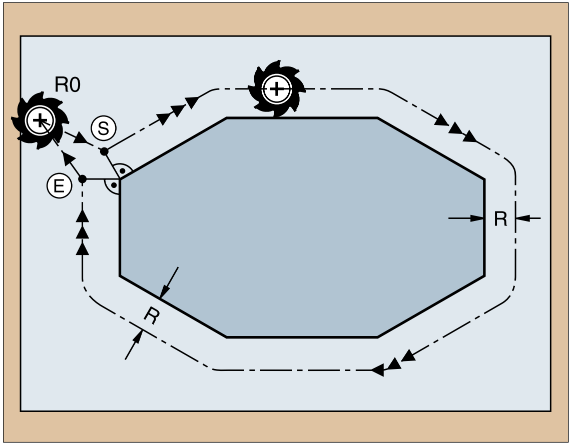
▶ Execution of a positioning block with R0

Working **without radius compensation** (e.g. drilling):

▶ Tool movement with R0



▼ Ⓢ = Start; Ⓔ = End

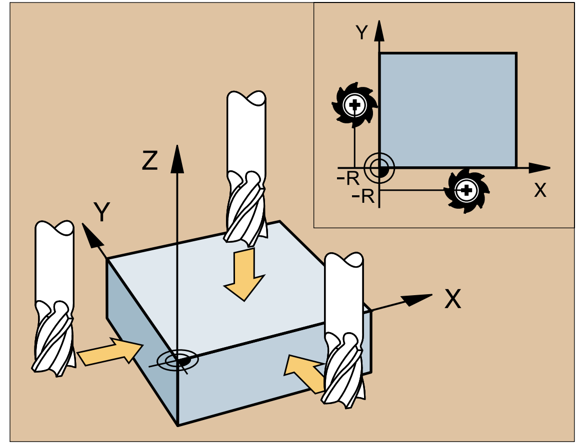


Fundamentals

Datum Setting without a 3-D Touch Probe

During datum setting you set the TNC display to the coordinates of a known position on the workpiece:

- Insert a zero tool with known radius
- Select the manual operation or electronic handwheel mode
- Touch the reference surface in the tool axis with the tool and enter its length
- Touch the reference surface in the working plane with the tool and enter the position of the tool center

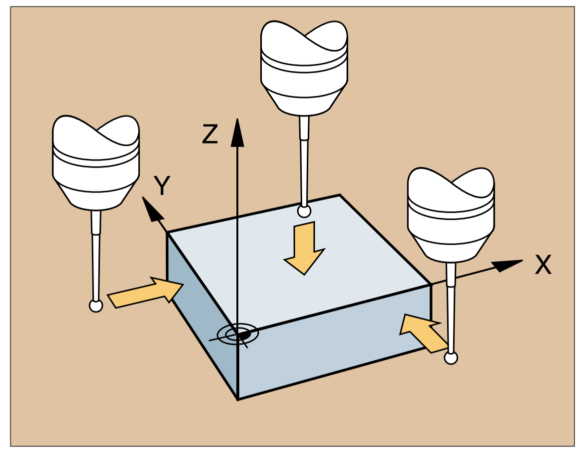


Setup and Measurement with 3-D Touch Probes

A HEIDENHAIN 3-D touch probe enables you to setup the machine very quickly, simply and precisely.

Besides the probing functions for workpiece setup on the Manual and Electronic Handwheel modes, the Program Run modes provide a series of measuring cycles (see also the User's Manual for Touch Probe Cycles):

- Measuring cycles for measuring and compensating workpiece misalignment
- Measuring cycles for automatic datum setting
- Measuring cycles for automatic workpiece measurement with tolerance checking and automatic tool compensation



Contour Approach and Departure

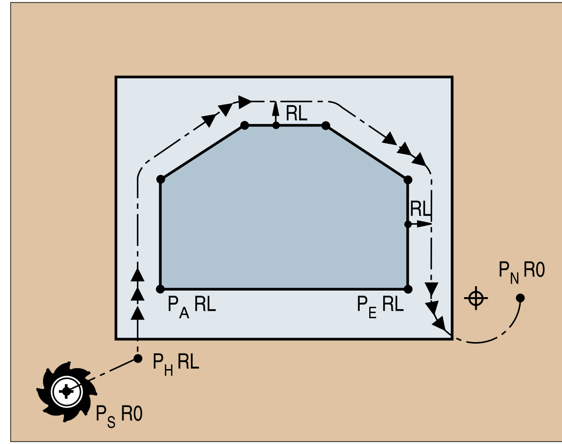
Starting point P_S
P_S lies outside of the contour and must be approached without radius compensation.

Auxiliary point P_H
P_H lies outside of the contour and is calculated by the TNC.

> The tool moves from the starting point P_S to the auxiliary point P_H at the feed rate last programmed feed rate!

First contour point P_A and last contour point P_E
The first contour point P_A is programmed in the APPR (approach) block. The last contour point is programmed as usual.

End point P_N
P_N lies outside of the contour and results from the DEP (departure) block. P_N is automatically approached with R0.



Path Functions for Approach and Departure

APPR DEP

▶ Press the soft key with the desired path function:

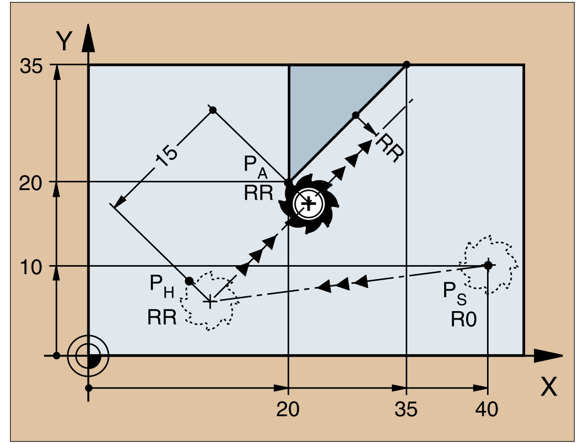
APPR LT	DEP LT	Straight line with tangential connection
APPR LN	DEP LN	Straight line perpendicular to the contour point
APPR CT	DEP CT	Circular arc with tangential connection
APPR LCT	DEP LCT	Straight line segment tangentially connected to the contour through an arc

- Program a radius compensation in the APPR block!
- DEP blocks set the radius compensation to 0!

Approaching on a Straight Line with Tangential Connection

- Coordinates for the first contour point P_A
- Distance Len (length) from P_H to P_A
  Enter a length Len > 0
- Tool radius compensation RR/RL

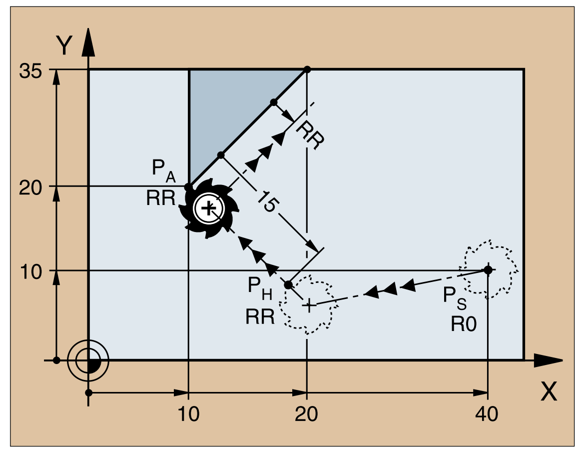
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR LT X+20 Y+20 LEN 15 RR F100
9 L X+35 Y+35
```



Approaching on a Straight Line Perpendicular to the First Contour Element

- Coordinates for the first contour point P_A
- Distance Len (length) from P_H to P_A
  Enter a length Len > 0
- Tool radius compensation RR/RL

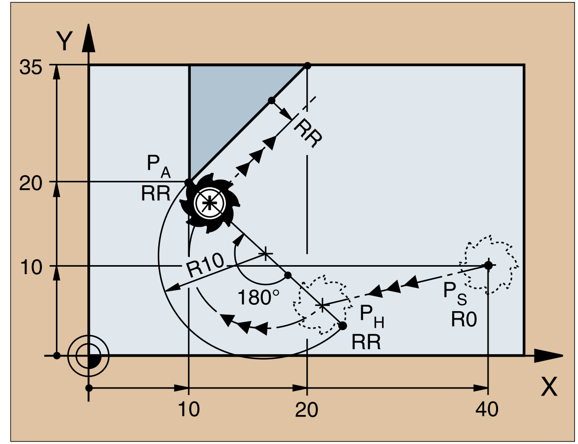
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR LN X+10 Y+20 LEN 15 RR F100
9 L X+20 Y+35
```



Approaching Tangentially on an Arc

- Coordinates for the first contour point P_A
- Radius R
  Enter a radius R > 0
- Circle center angle (CCA)
  Enter a CCA > 0
- Tool radius compensation RR/RL

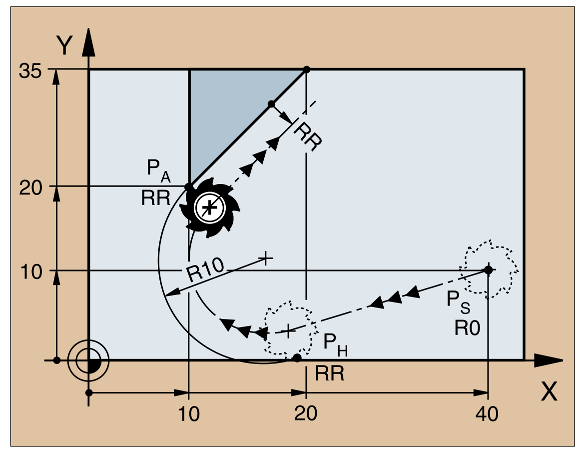
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR CT X+10 Y+20 CCA 180 R10 RR F100
9 L X+20 Y+35
```



Approaching Tangentially on an Arc and a Straight Line

- Coordinates for the first contour point P_A
- Radius R
  Enter a radius R > 0
- Tool radius compensation RR/RL

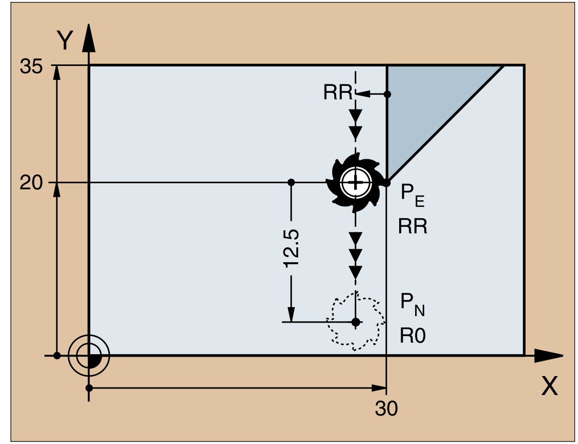
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR LCT X+10 Y+20 R10 RR F100
9 L X+20 Y+35
```



Departing Tangentially on a Straight Line

- Distance Len (length) from P_E to P_N
  Enter a length Len > 0

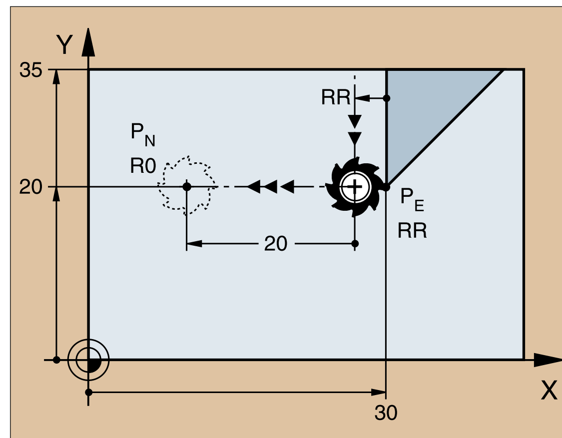
```
23 L X+30 Y+35 RR F100
24 L Y+20 RR F100
25 DEP LT LEN 12.5 F100 M2
```



Departing on a Straight Line Perpendicular to the Last Contour Element

- Distance Len (length) from P_E to P_N
  Enter a length Len > 0

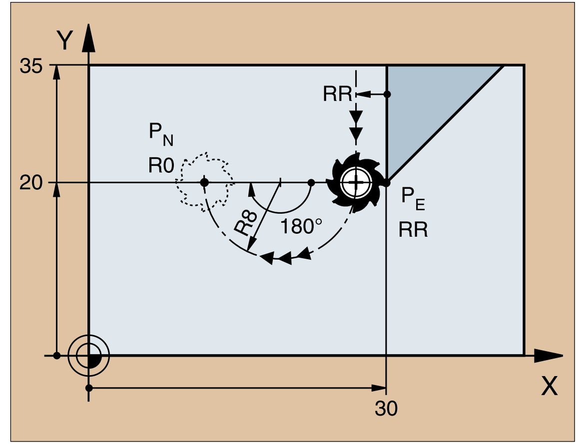
```
23 L X+30 Y+35 RR F100
24 L Y+20 RR F100
25 DEP LN LEN+20 F100 M2
```



Departing Tangentially on an Arc

- Radius R
  Enter a radius R > 0
- Circle center angle (CCA)

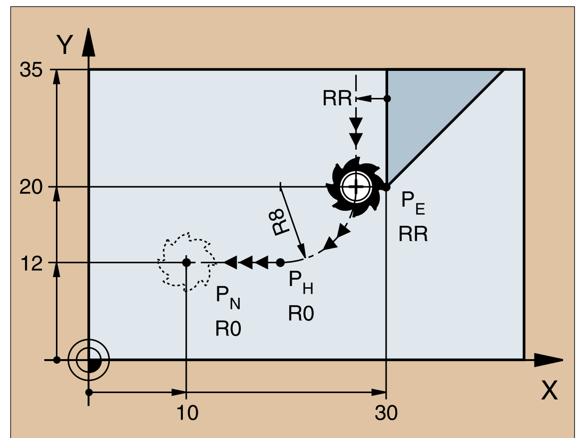
```
23 L X+30 Y+35 RR F100
24 L Y+20 RR F10
25 DEP CT CCA 180 R+8 F100 M2
```



Departing on an Arc Tangentially Connecting the Contour and a Straight Line

- Coordinates of the end point P_N
- Radius R
  Enter a radius R > 0

```
23 L X+30 Y+35 RR F100
24 L Y+20 RR F100
25 DEP LCT X+10 Y+12 R8 F100 M2
```



Path Functions for Positioning Blocks

See „Programming: Programming contours“.

Programming the Direction of Traverse

Regardless of whether the tool or the workpiece is actually moving, you always program as if the tool is moving and the workpiece is stationary.

Entering the Target Positions

Target positions can be entered in Cartesian or polar coordinates – either as absolute or incremental values, or with both absolute and incremental values in the same block.

Entries in the Positioning Block

A complete positioning block contains the following data:

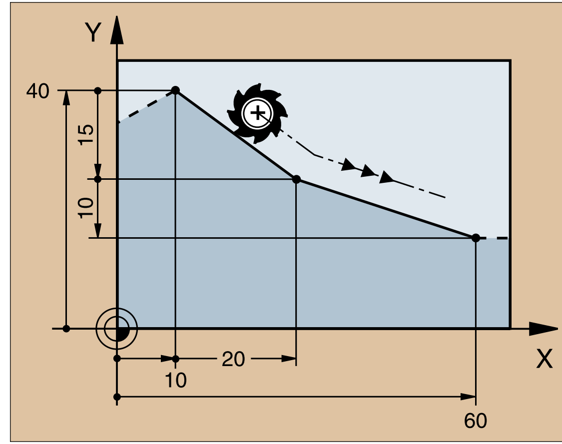
- Path function
- Coordinates of the contour element end points (target position)
- Radius compensation RR/RL/R0
- Feed rate F
- Miscellaneous function M

Before you execute a part program, always pre-position the tool to prevent the possibility of damaging the tool or workpiece!

Path functions		
Straight line	L	Page **19**
Chamfer between two straight lines	CHF	Page **20**
Corner rounding	RND	Page **20**
Circle center or **pole for polar coordinates**	CC	Page **21**
Circular path around the circle center CC	C	Page **21**
Circular path with known radius	CR	Page **22**
Circular path with tangential connection to previous contour	CT	Page **23**
FK Free Contour Programming	FK	Page **25**

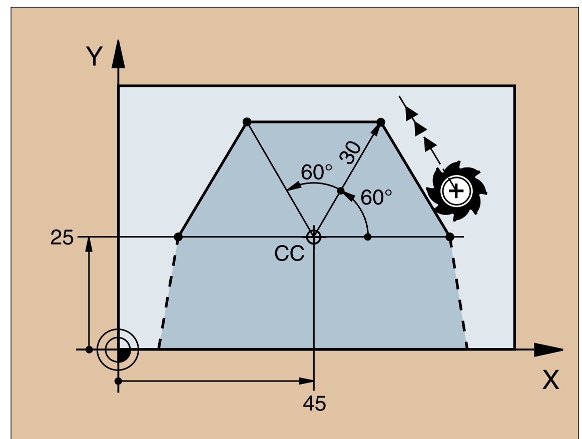
Straight Line

- Coordinates of the straight line end point
- Tool radius compensation RR/RL/R0
- Feed rate F
- Miscellaneous function M



With Cartesian coordinates:

```
7 L X+10 Y+40 RL F200 M3
8 L IX+20 IY-15
9 L X+60 IY-10
```



With polar coordinates:

```
12 CC X+45 Y+25
13 LP PR+30 PA+0 RR F300 M3
14 LP PA+60
15 LP IPA+60
16 LP PA+180
```

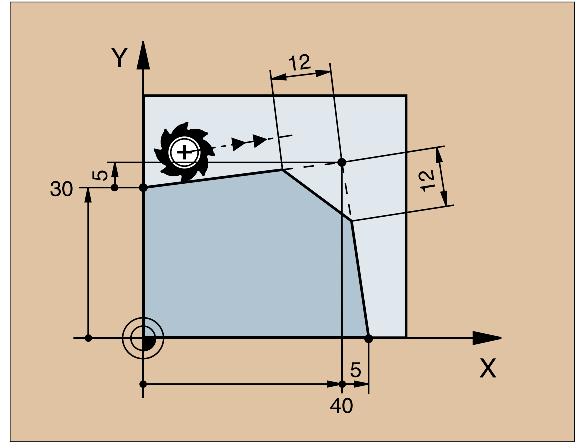
- You must first define the pole CC before you can program polar coordinates!
- Program the pole CC only in Cartesian coordinates!
- The pole CC remains effective until you define a new one!

Inserting a Chamfer Between Two Straight Lines

- Chamfer side length
- Feed rate F for the chamfer

```
7 L X+0 Y+30 RL F300 M3
8 L X+40 IY+5
9 CHF 12 F250
10 L IX+5 Y+0
```

- You cannot start a contour with a CHF block!
- The radius compensation before and after the CHF block must be the same!
- An inside chamfer must be large enough to accommodate the current tool!



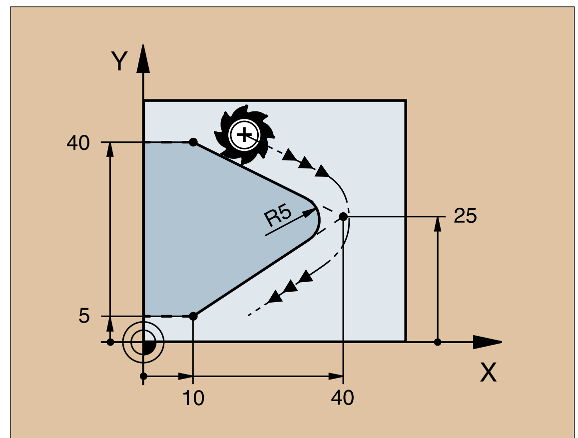
Corner Rounding

The beginning and end of the arc extend tangentially from the previous and subsequent contour elements.

- Radius R of the circular arc
- Feed rate F for corner rounding

```
5 L X+10 Y+40 RL F300 M3
6 L X+40 Y+25
7 RND R5 F100
8 L X+10 Y+5
```

An inside arc must be large enough to accommodate the current tool!



Circular Path Around the Circle Center CC

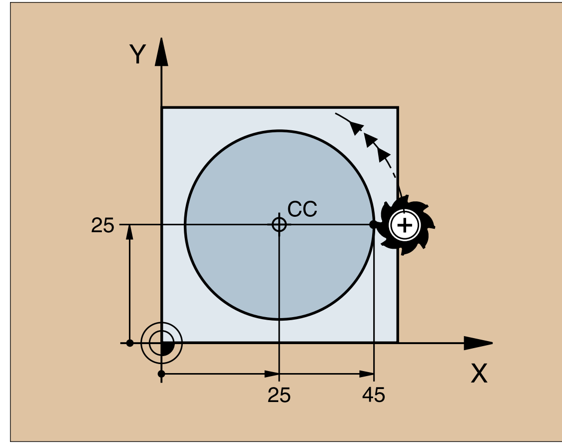
▶ Coordinates of the circle center CC

▶ Coordinates of the arc end point
▶ Direction of rotation DR

C and CP enable you to program a complete circle in one block.

With cartesian coordinates:

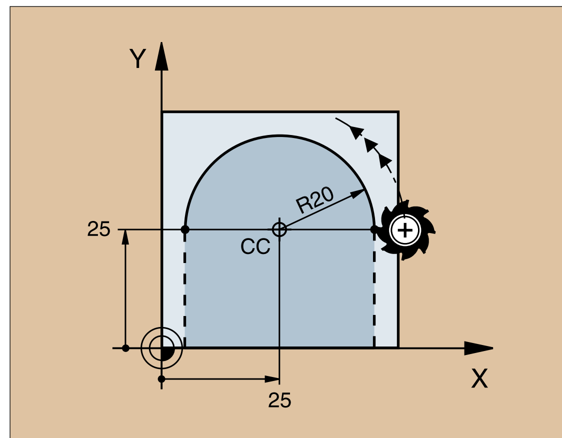
```
5 CC X+25 Y+25
6 L X+45 Y+25 RR F200 M3
7 C X+45 Y+25 DR+
```



With polar coordinates:

```
18 CC X+25 Y+25
19 LP PR+20 PA+0 RR F250 M3
20 CP PA+180 DR+
```

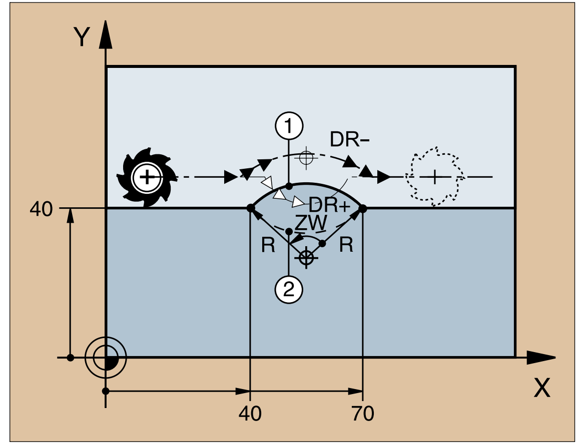
- Define the pole CC before programming polar coordinates!
- Program the pole CC only in Cartesian coordinates!
- The pole CC remains effective until you define a new one!
- The arc end point can be defined only with the polar coordinate angle (PA)!



Circular Path with Known Radius (CR)

- Coordinates of the arc end point
- Radius R
  If the central angle ZW > 180, R is negative.
  If the central angle ZW < 180, R is positive.
- Direction of rotation DR

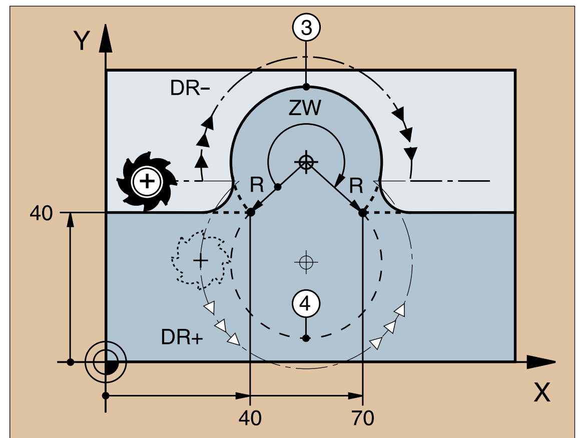
`10 L X+40 Y+40 RL F200 M3`	Arc starting point
`11 CR X+70 Y+40 R+20 DR-`	Arc ① or
`11 CR X+70 Y+40 R+20 DR+`	Arc ②



▲ Arcs ① and ②

▼ Arcs ③ and ④

`10 L X+40 Y+40 RL F200 M3`	Arc starting point
`11 CR X+70 Y+40 R-20 DR-`	Arc ③ or
`11 CR X+70 Y+40 R-20 DR+`	Arc ④

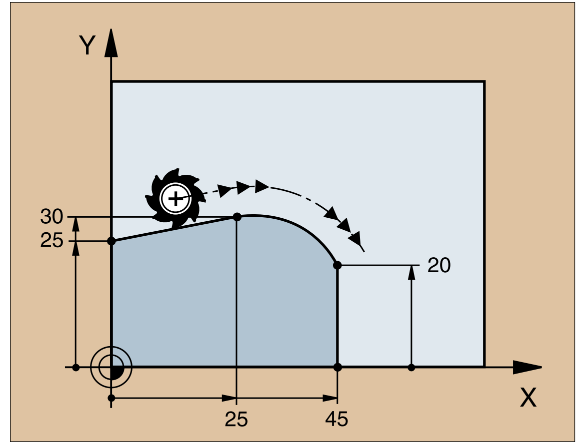


Circular Path CT with Tangential Connection

- Coordinates of the arc end point
- Radius compensation RR/RL/R0
- Feed rate F
- Miscellaneous function M

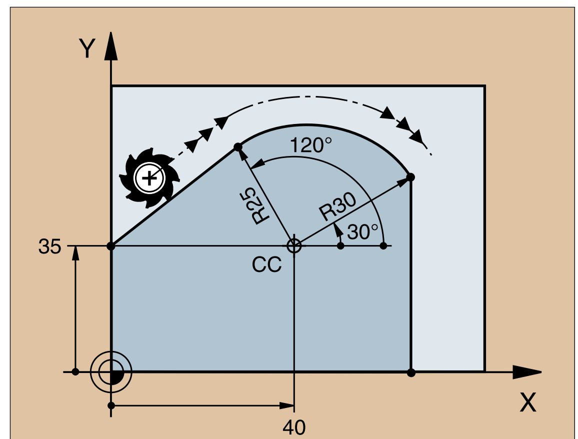
With cartesian coordinates:

```
5 L X+0 Y+25 RL F250 M3
6 L X+25 Y+30
7 CT X+45 Y+20
8 L Y+0
```



With polar coordinates:

```
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0
```

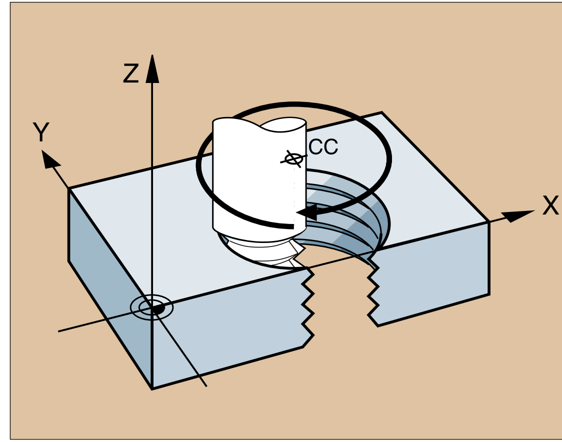


- Define the pole CC before programming polar coordinates!
- Program the pole CC only in Cartesian coordinates!
- The pole CC remains effective until you define a new one!

Helix (Only in Polar Coordinates)

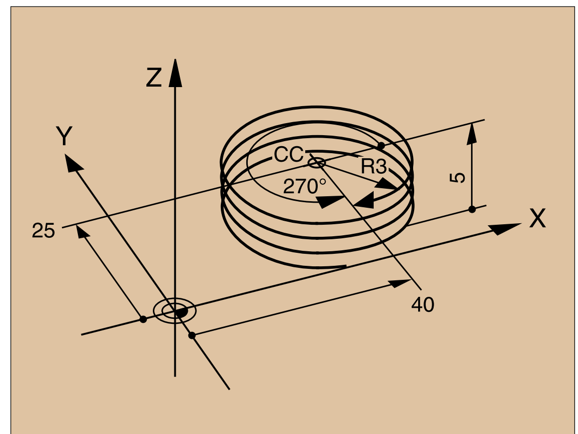
Calculations (upward milling direction)

Path revolutions:	n	=	Thread revolutions + overrun at start and end of thread
Total height:	h	=	Pitch P x path revolutions n
Incr. coord. angle:	IPA	=	Path revolutions n x 360°
Start angle:	PA	=	Angle at start of thread + angle for overrun
Start coordinate:	Z	=	Pitch P x (thread revolutions + thread overrun at start of thread)



Shape of helix

Internal thread	Work direction	Direction	Radius comp.
Right-hand	Z+	DR+	RL
Left-hand	Z+	DR–	RR
Right-hand	Z–	DR–	RR
Left-hand	Z–	DR+	RL
External thread			
Right-hand	Z+	DR+	RR
Left-hand	Z+	DR–	RL
Right-hand	Z–	DR–	RL
Left-hand	Z–	DR+	RR



M6 x 1 mm thread with 5 revolutions:

```
12 CC X+40 Y+25
13 L Z+0 F100 M3
14 LP PR+3 PA+270 RL
15 CP IPA-1800 IZ+5 DR- RL F50
```

FK Free Contour Programming

See "Programming Tool Movements – FK Free Contour Programming"

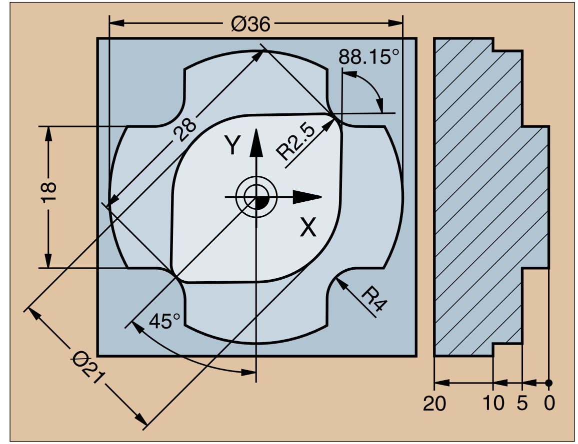
If the end point coordinates are not given in the workpiece drawing or if the drawing gives dimensions that cannot be entered with the gray path function keys, you can still program the part by using the "FK Free Contour Programming."

Possible data on a contour element:

- Known coordinates of the end point
- Auxiliary points on the contour element
- Auxiliary points near the contour element
- A reference to another contour element
- Directional data (angle) / position data
- Data regarding the course of the contour

To use FK programming properly:

- All contour elements must lie in the working plane.
- Enter all available data on each contour element.
- If a program contains both FK and conventional blocks, the FK contour must be fully defined before you can return to conventional programming.



▲ These dimensions can be programmed with FK

Working with the Interactive Graphics

Select the PGM+GRAPHICS screen layout!

The interactive graphics show the contour as you are programming it. If the data you enter can apply to more than one solution, the following soft keys will appear:

SHOW SOLUTION	To show the possible solutions
SELECT SOLUTION	To enter the displayed solution in the part program
END SELECT	To enter data for subsequent contour elements
START SINGLE	To graphically display the next programmed block

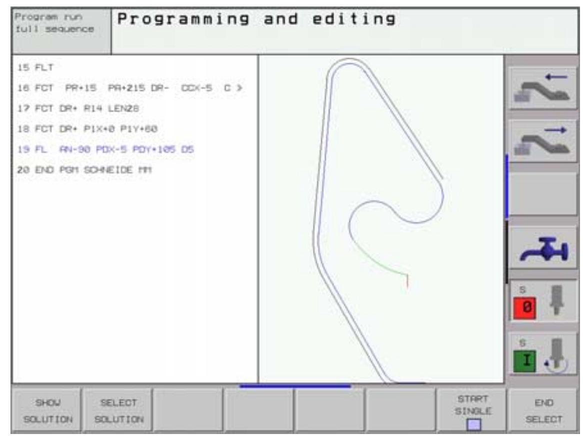
Standard colors of the interactive graphics

Fully defined contour element

The displayed element is one of a limited number of possible solutions

The element is one of an infinite number of solutions

Contour element from a subprogram



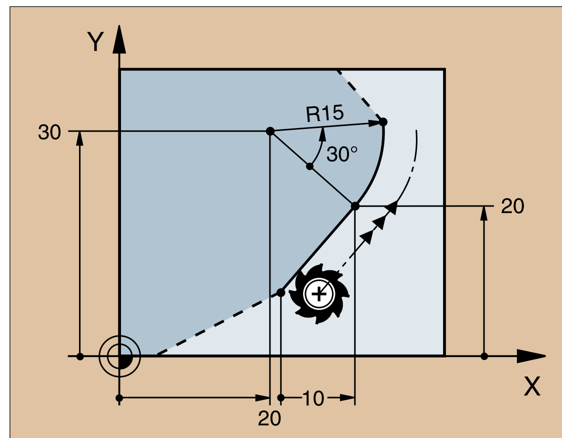
Initiating the FK Dialog

FK	Initiate the FK dialog
Straight Circular	
FL FC	Contour element without tangential connection
FLT FCT	Contour element with tangential connection
FPOL	Pole for FK programming

End Point Coordinates X, Y or PA, PR

X Y	Cartesian coordinates X and Y
PR PA	Polar coordinates referenced to FPOL
I	Incremental input

```
7 FPOL X+20 Y+30
8 FL IX+10 Y+20 RR F100
9 FCT PR+15 IPA+30 DR+ R15
```



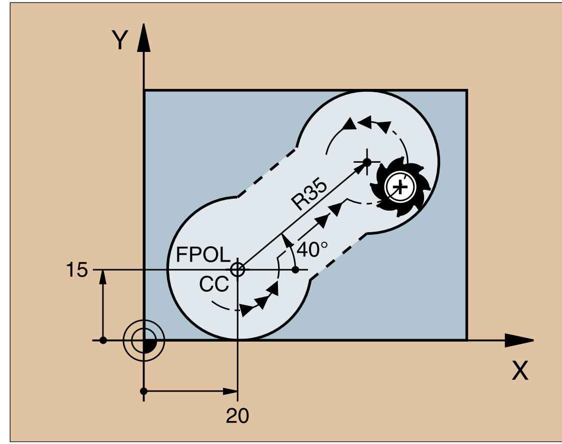
Circle Center (CC) in an FC/FCT block

Cartesian coordinates of the circle center

Polar coordinates of the circle center referenced to FPOL

Incremental input

```
10 FC CCX+20 CCY+15 DR+ R15
11 FPOL X+20 Y+15
...
13 FC DR+ R15 CCPR+35 CCPA+40
```



Auxiliary Points

... P1, P2, P3 on a contour

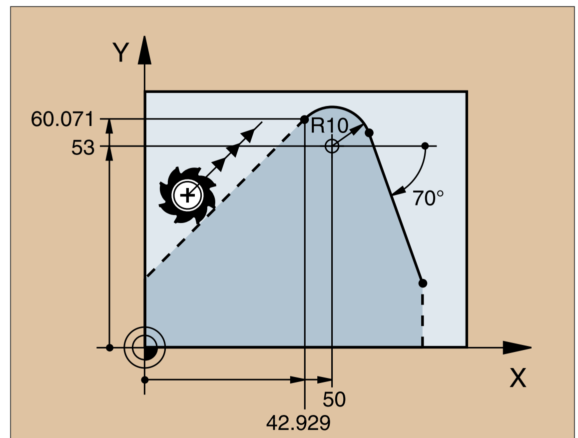
For straight lines: up to 2 auxiliary points
For circles: up to 3 auxiliary points

... next to a contour

Coordinates of the auxiliary points

Perpendicular distance

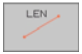
```
13 FC DR- R10 P1X+42.929 P1Y+60.071
14 FLT AN-70 PDX+50 PDY+53 D10
```



Direction and Length of the Contour Element

Data on a straight line

Gradient angle of a straight line



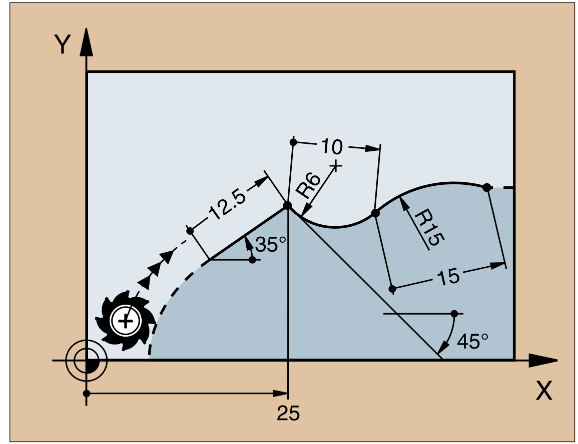
Length of a straight line

Data on a circular path

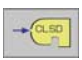
Gradient angle of the entry tangent

Length of an arc chord

```
27 FLT X+25 LEN 12.5 AN+35 RL F200
28 FC DR+ R6 LEN 10 AN-45
29 FCT DR- R15 LEN 15
```

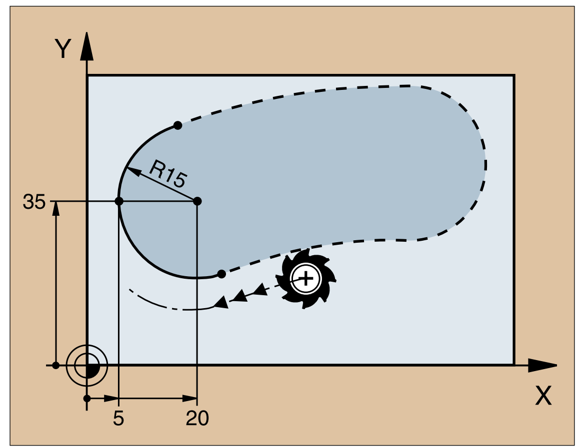


Identifying a closed contour

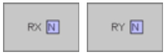


Beginning: CLSD+
End: CLSD–

```
12 L X+5 Y+35 RL F500 M3
13 FC DR- R15 CLSD+ CCX+20 CCY+35
...
17 FCT DR- R+15 CLSD-
```



Values Relative to Block N: Entering Coordinates

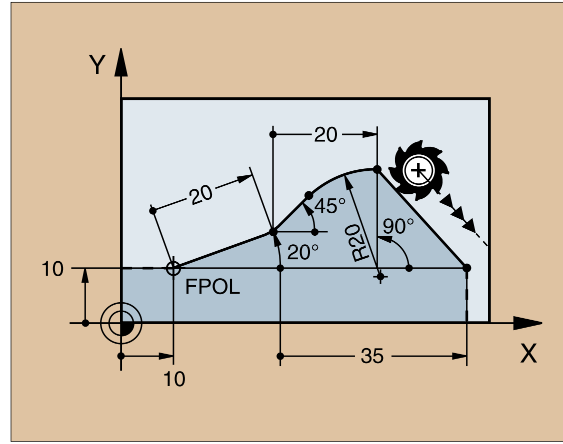


Cartesian coordinates relative to block N

Polar coordinates relative to block N

- Relative data must be entered incrementally!
- CC can also be programmed in relative values!

```
12 FPOL X+10 Y+10
13 FL PR+20 PA+20
14 FL AN+45
15 FCT IX+20 DR- R20 CCA+90 RX 13
16 FL IPR+35 PA+0 RPR 13
```



Values Relative to Block N: Direction and Distance of the Contour Element

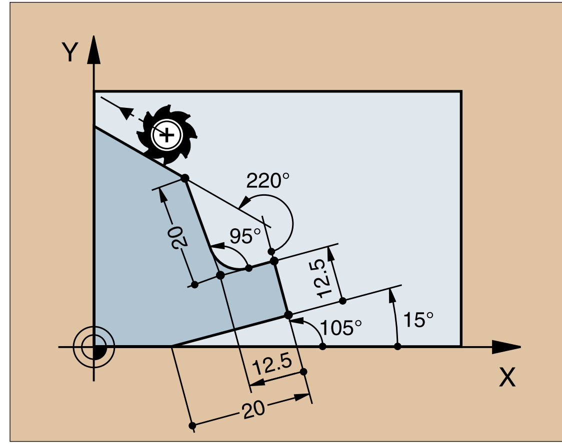
Gradient angle

Parallel to a straight contour element
Parallel to the entry tangent of an arc

Distance from a parallel element

Always enter relative values incrementally!

```
17 FL LEN 20 AN+15
18 FL AN+105
19 FL LEN 12.5 PAR 17 DP 12.5
20 FSELECT 2
21 FL LEN 20 IAN+95
22 FL IAN+220 RAN 18
```



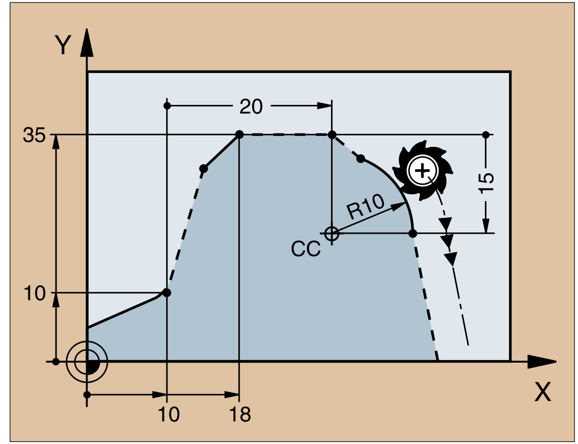
Values Relative to Block N: Circle Center CC

Cartesian coordinates of a circle center relative to block N

Polar coordinates of the circle center relative to block N

Always enter relative data as incremental values!

```
12 FL X+10 Y+10 RL
13 FL ...
14 FL X+18 Y+35
15 FL ...
16 FL ...
17 FC DR- R10 CCA+0 ICCX+20 ICCY-15
   RCCX12 RCCY14
```



Subprograms and Program Section Repeats

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as needed.

Working with Subprograms

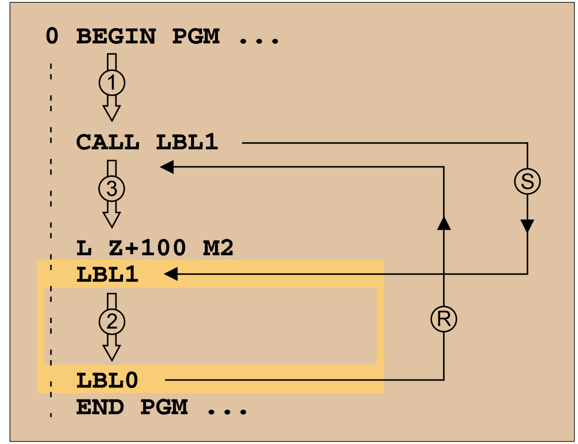
① The main program runs up to the subprogram call CALL LBL1.

② The subprogram–labeled with LBL1–runs through to its end LBL0.

③ The main program resumes.

It's good practice to place subprograms after the main program end (M2).

- Answer the dialog prompt REP with the NOENT key!
- You cannot call LBL0!



Ⓢ = Jump; Ⓡ = Return jump

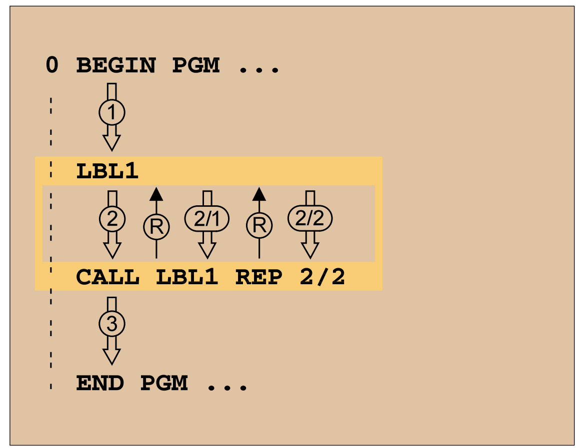
Working with Program Section Repeats

① The main program runs up to the call for a section repeat CALL LBL1 REP2/2.

② The program section between LBL1 and CALL LBL1 REP2/2 is repeated the number of times indicated with REP.

③ After the last repetition the main program resumes.

Altogether, the program section is run once more than the number of programmed repeats!

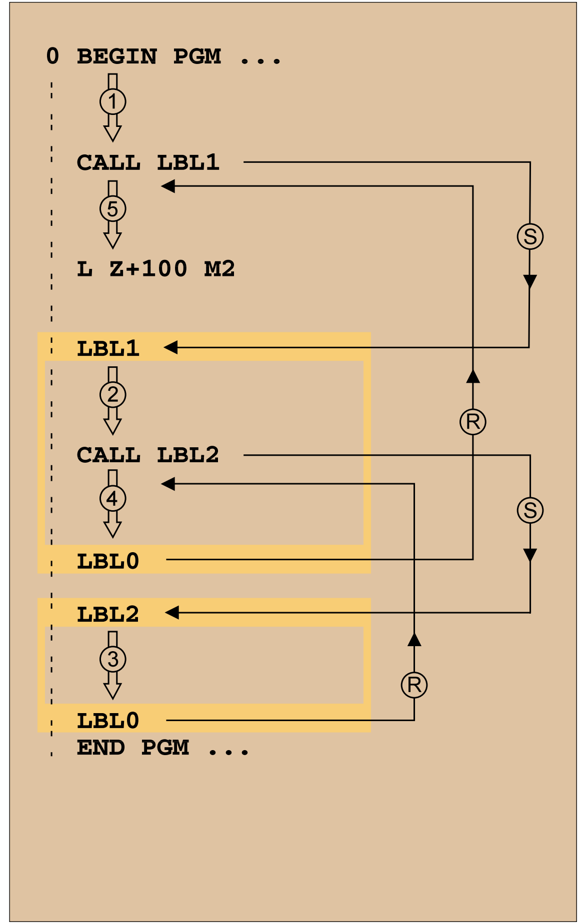


Subprograms

Subprogram Nesting: A Subprogram within a Subprogram

① The main program runs up to the first subprogram call CALL LBL1.
② Subprogram 1 runs up to the second subprogram call CALL LBL2.
③ Subprogram 2 runs to its end.
④ Subprogram 1 resumes and runs to its end.
⑤ The main program resumes.

- A subprogram cannot call itself!
- Subprograms can be nested up to a maximum depth of 8 levels!



Ⓢ = Jump; Ⓡ = Return jump ▶

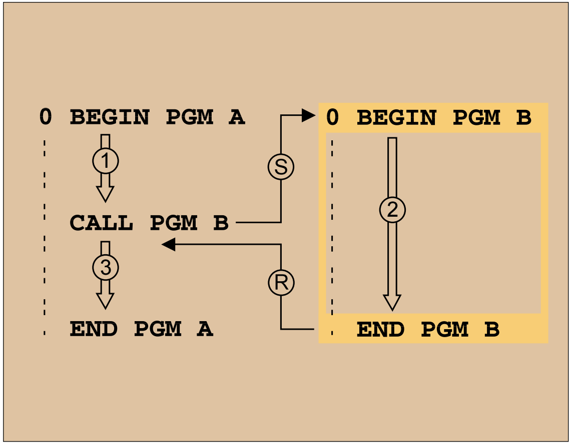
Any Program as a Subprogram

① The calling program A runs up to the program call CALL PGM B.

② The called program B runs through to its end.

③ The calling program A resumes.

> The **called** program must not end with M2 or M30!



▲ Ⓢ = Jump; Ⓡ = Return jump

Working with Cycles

Certain frequently needed machining sequences are stored in the TNC as cycles. Coordinate transformations and some special functions are also available as cycles.

- In a cycle, positioning data entered in the tool axis are always incremental, even without the I key!
- The algebraic sign of the cycle parameter depth determines the working direction!

Example

```
6 CYCL DEF 1.0 PECKING
7 CYCL DEF 1.1 SET UP 2
8 CYCL DEF 1.2 DEPTH -15
9 CYCL DEF 1.3 PECKG 10
...
```

Feed rates are entered in mm/min, the dwell time in seconds.

Defining cycles

▶ Select the Cycle Overview:

▶ Select the cycle group

▶ Select the cycle

Cycles for Machining Holes and Threads

1	PECKING	Page 39
200	DRILLING	Page 40
201	REAMING	Page 41
202	BORING	Page 42
203	UNIVERSAL DRILLING	Page 43
204	COUNTERBORE BACK	Page 44
205	UNIVERSAL PECKING	Page 45
208	BORE MILLING	Page 46
2	TAPPING	Page 47
206	TAPPING NEW	Page 48
17	RIGID TAPPING	Page 48
207	RIGID TAPPING NEW	Page 49
18	THREAD CUTTING	Page 49
209	TAPPING W/ CHIP BRKG	Page 50
262	THREAD MILLING	Page 51
263	THREAD MLLNG/CNTSNKG	Page 52
264	THREAD DRILLNG/MLLNG	Page 53
265	HEL. THREAD DRLG/MLG	Page 54
267	OUTSIDE THREAD MLLNG	Page 55

Continued on next page ▶

Pockets, Studs, and Slots

Point Patterns

SL Cycles

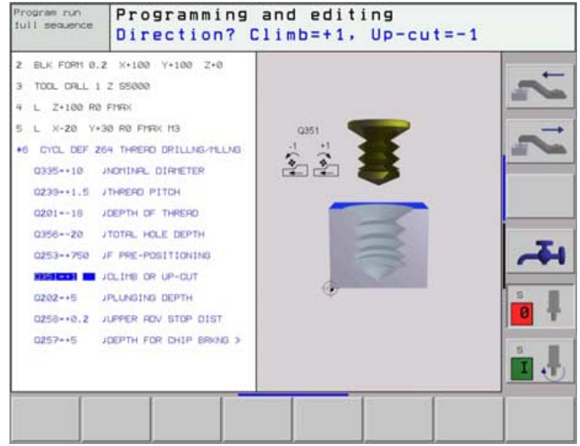
Cycles for Multipass Milling

Cycles for Coordinate Transformations

Spezial Cycles

Graphic Support During Cycle Programming

As you create a program, the TNC provides you with graphic illustrations of the input parameters.



Calling a Cycle

The following cycles are effective as soon as they are defined:

- Cycles for coordinate transformations
- DWELL TIME cycle
- The SL cycles CONTOUR GEOMETRY and CONTOUR DATA
- Point patterns
- TOLERANCE cycle

All other cycles go into effect when they are called through

- CYCL CALL: effective for one block
- CYCL CALL PAT: used non-modally in connection with point tables
- M99: effective for one block
- M89: effective until canceled (depends on machine parameter settings)

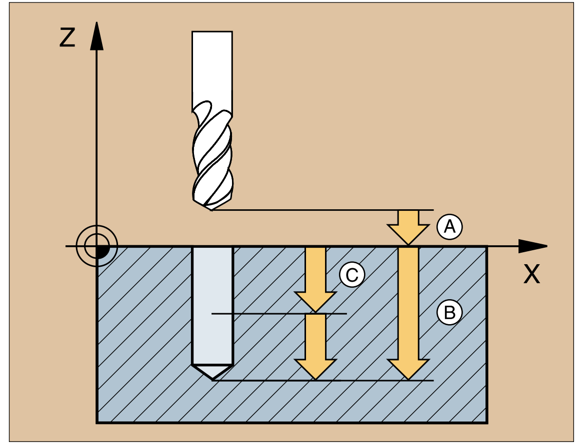
Cycles for Machining Holes and Threads

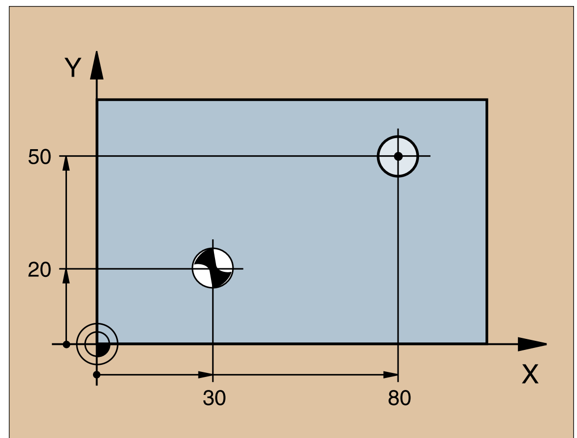
PECKING (1)

- CYCL DEF: Select Cycle 1 PECKING
  - Set-up clearance: Ⓐ
  - Total hole depth (distance from the workpiece surface to the bottom of the hole): Ⓑ
  - Pecking depth: Ⓒ
  - Dwell time in seconds
  - Feed rate F

If the pecking depth is greater than or equal to the total hole depth, the tool drills the entire hole in one plunge.

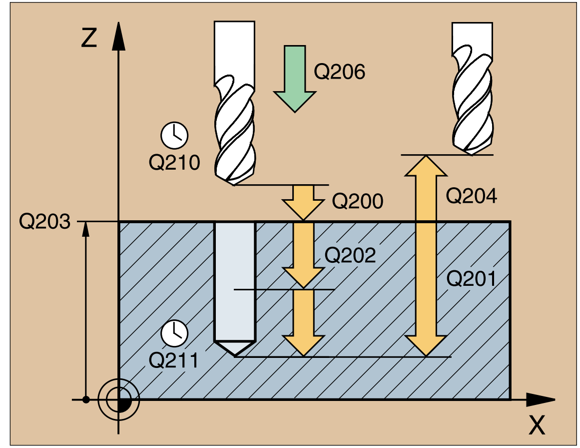
```
6 CYCL DEF 1.0 PECKING
7 CYCL DEF 1.1 SET UP +2
8 CYCL DEF 1.2 DEPTH -15
9 CYCL DEF 1.3 PECKG +7.5
10 CYCL DEF 1.4 DWELL 1
11 CYCL DEF 1.5 F80
12 L Z+100 R0 FMAX M6
13 L X+30 Y+20 FMAX M3
14 L Z+2 FMAX M99
15 L X+80 Y+50 FMAX M99
16 L Z+100 FMAX M2
```





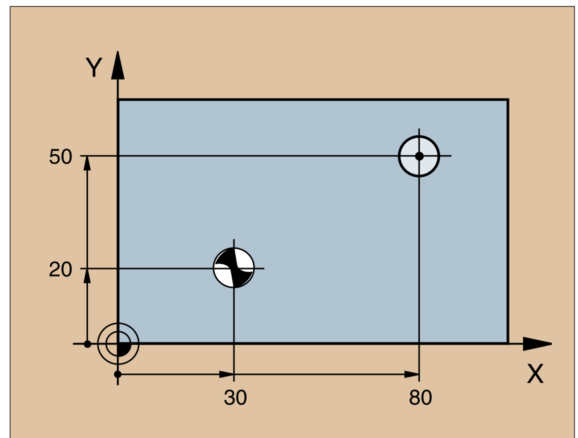
DRILLING (200)

- CYCL DEF: Select Cycle 200 DRILLING
  - Set-up clearance: Q200
  - Depth – Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Pecking depth: Q202
  - Dwell time at top: Q210
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Dwell time at depth: Q211



The TNC automatically pre-positions the tool in the tool axis. If the pecking depth is greater than or equal to the depth, the tool drills to the depth in one plunge.

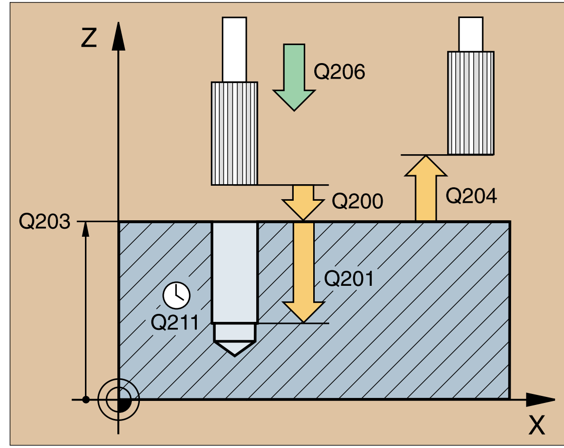
```
11 CYCL DEF 200 DRILLING
   Q200 = 2     ;SET-UP CLEARANCE
   Q201 = -15   ;DEPTH
   Q206 = 250   ;FEED RATE FOR PLUNGING
   Q202 = 5     ;PLUNGING DEPTH
   Q210 = 0     ;DWELL TIME AT TOP
   Q203 = +0    ;SURFACE COORDINATE
   Q204 = 100   ;2ND SET-UP CLEARANCE
   Q211 = 0.1   ;DWELL TIME AT DEPTH
12 L Z+100 R0 FMAX M6
13 L X+30 Y+20 FMAX M3
14 CYCL CALL
15 L X+80 Y+50 FMAX M99
16 L Z+100 FMAX M2
```



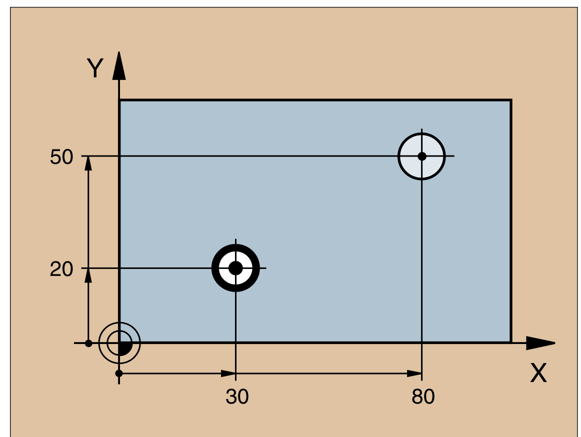
REAMING (201)

- CYCL DEF: Select Cycle 201 REAMING
  - Set-up clearance: Q200
  - Depth—Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Dwell time at depth: Q211
  - Retraction feed rate: Q208
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204

The TNC automatically pre-positions the tool in the tool axis.



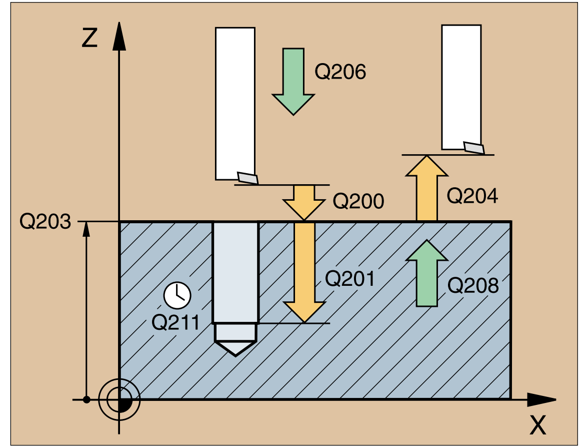

```
11 CYCL DEF 201 REAMING
   Q200 = 2     ;SET-UP CLEARANCE
   Q201 = -15   ;DEPTH
   Q206 = 100   ;FEED RATE FOR PLNGNG
   Q211 = 0.5   ;DWELL TIME AT DEPTH
   Q208 = 250   ;RETRACTION FEED RATE
   Q203 = +0    ;SURFACE COORDINATE
   Q204 = 100   ;2ND SET-UP CLEARANCE
12 L Z+100 R0 FMAX M6
13 L X+30 Y+20 FMAX M3
14 CYCL CALL
15 L X+80 Y+50 FMAX M99
16 L Z+100 FMAX M2
```



BORING (202)

- The machine and TNC must be prepared for the BORING cycle by the machine tool builder!
- This cycle requires a position-controlled spindle!

Danger of collision! Choose a disengaging direction that moves the tool away from the wall of the hole.

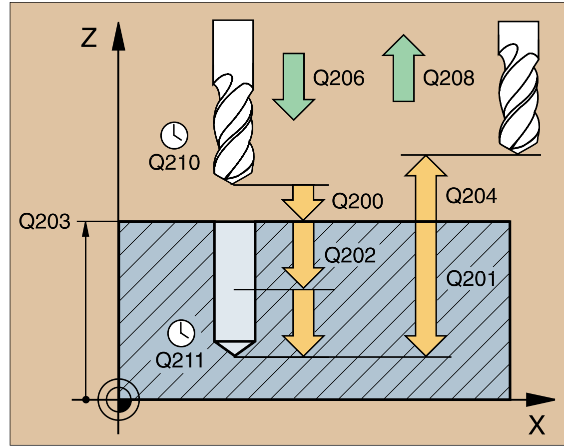


- CYCL DEF: Select Cycle 202 BORING
  - Set-up clearance: Q200
  - Depth—Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Dwell time at depth: Q211
  - Retraction feed rate: Q208
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Disengaging directn (0/1/2/3/4) at bottom of hole: Q214
  - Angle for oriented spindle stop: Q336

The TNC automatically pre-positions the tool in the tool axis.

UNIVERSAL DRILLING (203)

- CYCL DEF: Select Cycle 203 UNIVERSAL DRILLING
  - Set-up clearance: Q200
  - Depth—Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Pecking depth: Q202
  - Dwell time at top: Q210
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Decrement after each pecking depth: Q212
  - Nr of breaks—Number of chip breaks before retraction: Q213
  - Min. pecking depth if a decrement has been entered: Q205
  - Dwell time at depth: Q211
  - Retraction feed rate: Q208
  - Retract dist. for chip breaking: Q256

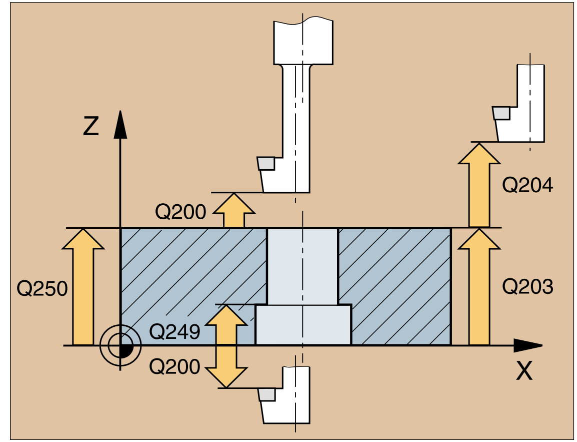


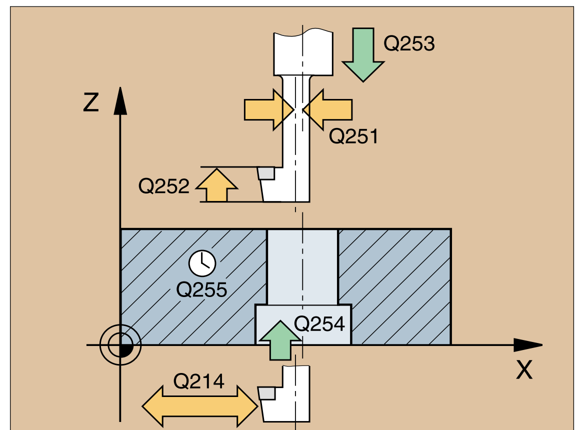
The TNC automatically pre-positions the tool in the tool axis. If the pecking depth is greater than or equal to the depth, the tool drills to the depth in one plunge.

COUNTERBORE BACK (204)

- The machine and TNC must be prepared for the COUNTERBORE BACK cycle by the machine tool builder!
- This cycle requires a position-controlled spindle!

- Danger of collision! Select the disengaging direction that gets the tool clear of the counterbore floor!
- Use this cycle only with a reverse boring bar!

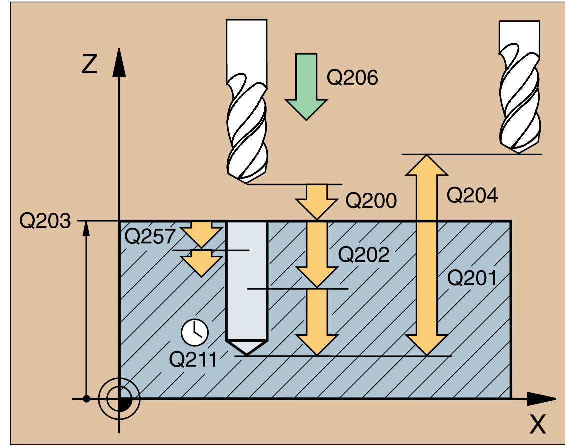




▶ CYCL DEF: Select Cycle 204 COUNTERBORE BACK
- ▶ Set-up clearance: Q200
- ▶ Depth of counterbore: Q249
- ▶ Material thickness: Q250
- ▶ Tool edge off-center distance: Q251
- ▶ Tool edge height: Q252
- ▶ Feed rate for pre-positioning: Q253
- ▶ Feed rate for counterboring: Q254
- ▶ Dwell time at counterbore floor: Q255
- ▶ Workpiece surface coordinate: Q203
- ▶ 2nd set-up clearance: Q204
- ▶ Disengaging direction (0/1/2/3/4): Q214
- ▶ Angle for oriented spindle stop: Q336

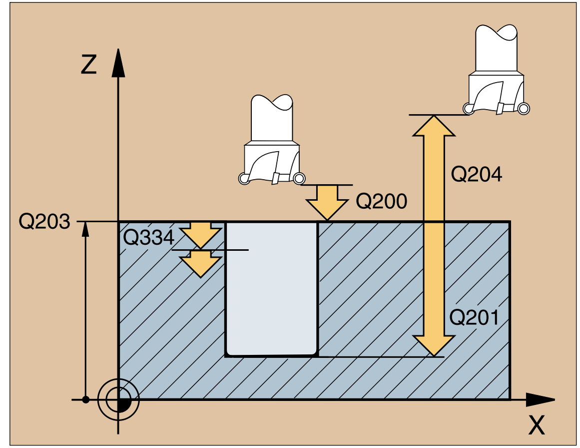
UNIVERSAL PECKING (205)

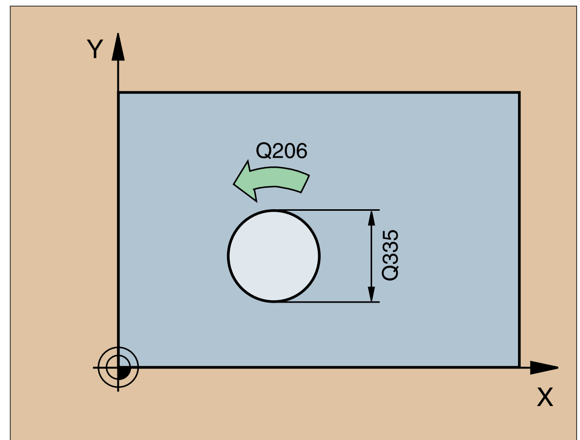
- CYCL DEF: Select Cycle 205 UNIVERSAL PECKING
  - Set-up clearance: Q200
  - Depth: Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Pecking depth: Q202
  - Workpiece surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Decrement after each pecking depth: Q212
  - Minimum pecking depth if decrement value entered: Q205
  - Upper advanced stop distance: Q258
  - Lower advanced stop distance: Q259
  - Infeed depth for chip breaking: Q257
  - Retract dist. for chip breaking: Q256
  - Dwell time at bottom: Q211



BORE MILLING (208)

- Pre-position to the center of the hole with R0
- CYCL DEF: Select Cycle 208 BORE MILLING
  - Set-up clearance: Q200
  - Depth: Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Infeed per helix: Q334
  - Workpiece surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Nominal diameter of hole: Q335
  - Pilot-drilled diameter: Q342

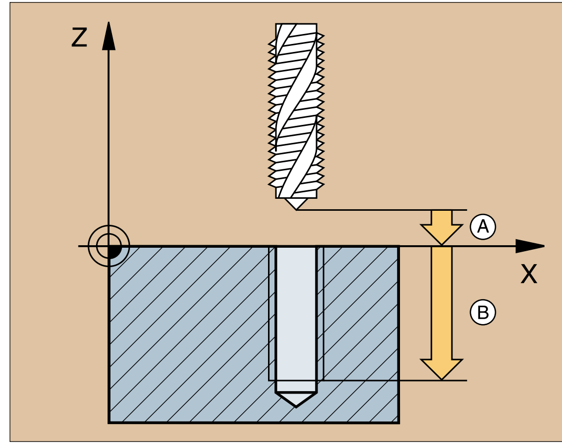




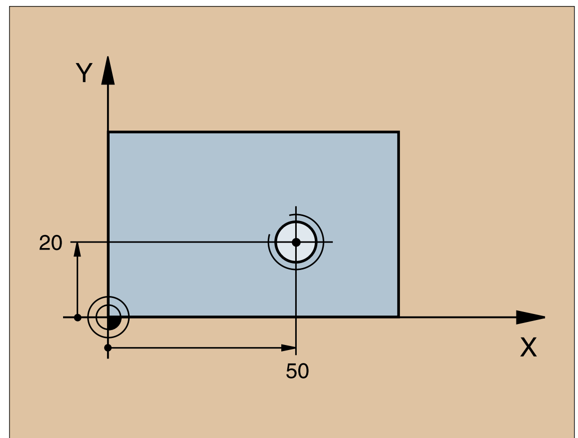
TAPPING (2) with Floating Tap Holder

- Insert the floating tap holder
- CYCL DEF: Select cycle 2 TAPPING
  - Set-up clearance: Ⓐ
  - Total hole depth (thread length = distance between the workpiece surface and the end of the thread): Ⓑ
  - Dwell time in seconds (a value between 0 and 0.5 seconds)
  - Feed rate F = Spindle speed S x thread pitch P

For tapping right-hand threads, actuate the spindle with M3, for left-hand threads use M4!



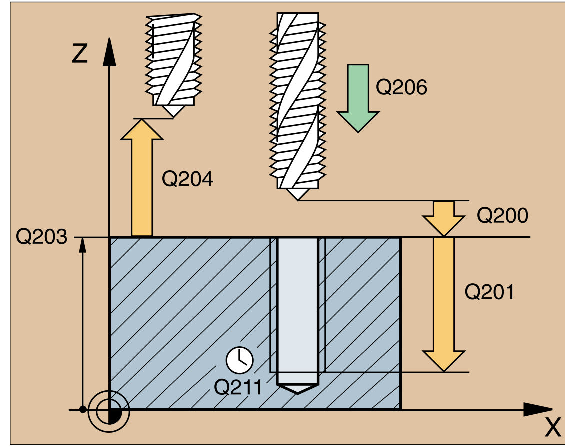

```
25 CYCL DEF 2.0 TAPPING
26 CYCL DEF 2.1 SET UP 3
27 CYCL DEF 2.2 DEPTH -20
28 CYCL DEF 2.3 DWELL 0.4
29 CYCL DEF 2.4 F100
30 L Z+100 R0 FMAX M6
31 L X+50 Y+20 FMAX M3
32 L Z+3 FMAX M99
```



TAPPING NEW (206) with Floating Tap Holder

- Insert the floating tap holder
- CYCL DEF: Select Cycle 206 TAPPING NEW
  - Set-up clearance: Q200
  - Depth: thread length = distance between the workpiece surface and the end of the thread: Q201
  - Feed rate F = spindle speed S x thread pitch P: Q206
  - Dwell time at bottom (enter a value between 0 and 0.5 seconds): Q211
  - Workpiece surface coordinate: Q203
  - 2nd set-up clearance: Q204

For tapping right-hand threads, actuate the spindle with M3, for left-hand threads use M4!

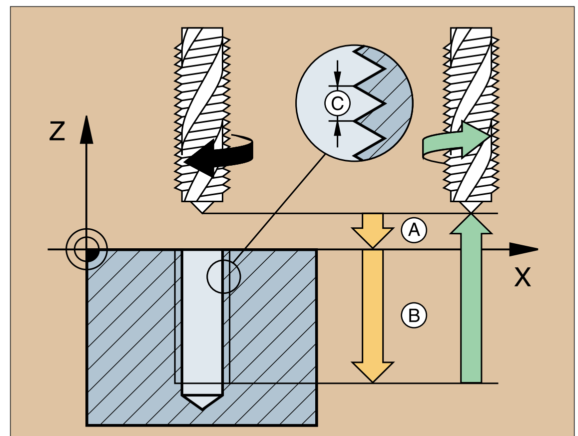


RIGID TAPPING (17) without Floating Tap Holder

- Machine and TNC must be prepared by the machine tool builder to perform rigid tapping!
- In rigid tapping, the spindle speed is synchronized with the tool axis feed rate!

- CYCL DEF: Select cycle 17 RIGID TAPPING
  - Set-up clearance: Ⓐ
  - Tapping depth (distance between workpiece surface and end of thread): Ⓑ
  - Pitch: Ⓒ

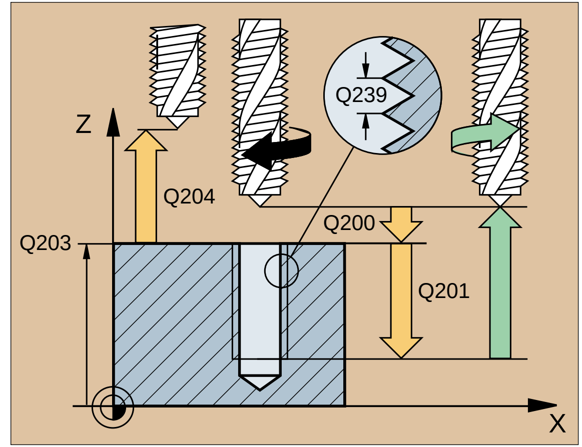
    The algebraic sign determines the direction of the thread:
    - Right-hand thread: +
    - Left-hand thread: –



RIGID TAPPING NEW (207) without Floating Tap Holder

- Machine and TNC must be prepared by the machine tool builder to perform rigid tapping!
- Rigid tapping is carried out with a controlled spindle!

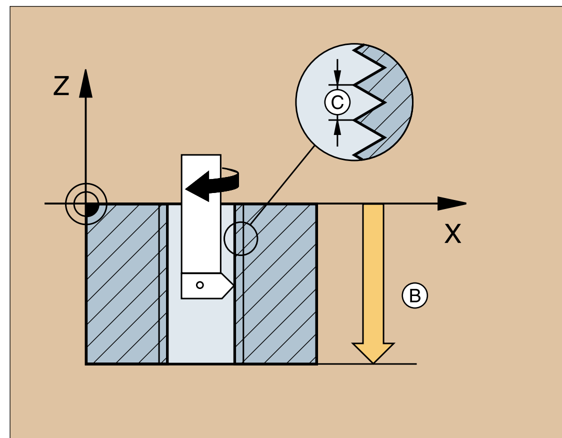
▶ CYCL DEF: Select Cycle 207 RIGID TAPPING NEW
  - ▶ Set-up clearance: Q200
  - ▶ Depth: thread length = distance between workpiece surface and end of thread: Q201
  - ▶ Pitch: Q239
    The algebraic sign determines the direction of the thread:
    - Right-hand thread: +
    - Left-hand thread: –
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204



THREAD CUTTING (18)

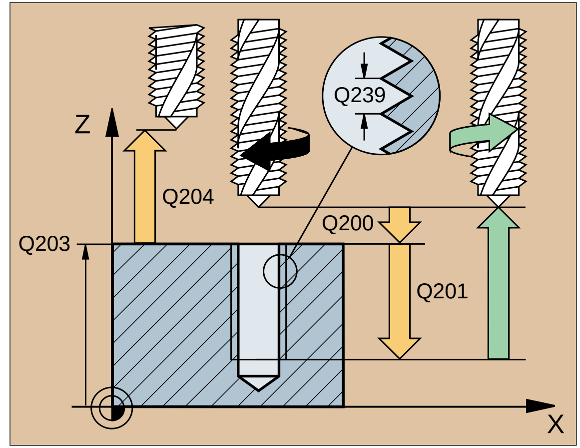
- The machine and TNC must be prepared by the machine tool builder for THREAD CUTTING!
- The spindle speed is synchronized with the tool axis feed rate!

▶ CYCL DEF: Select cycle 18 THREAD CUTTING
  - ▶ Depth (distance between workpiece surface and end of thread): Ⓑ
  - ▶ Pitch: Ⓒ
    The algebraic sign:
    - Right-hand thread: +
    - Left-hand thread: –



TAPPING WITH CHIP BREAKING (209)

- The machine and TNC must be prepared for the TAPPING WITH CHIP BREAKING cycle by the machine tool builder!
- This cycle requires a position-controlled spindle!



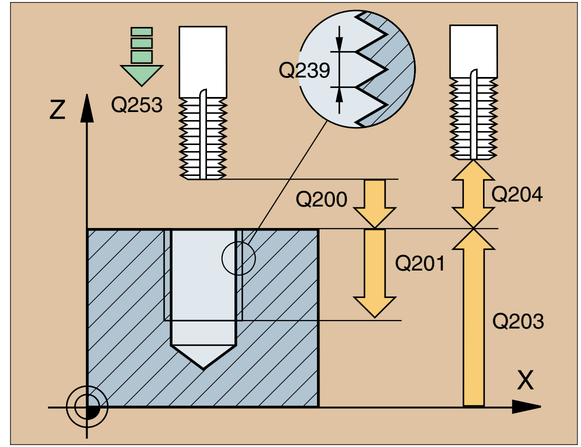
▶ CYCL DEF: Select Cycle 209 TAPPING W/ CHIP BRKG .
- ▶ Set-up clearance: Q200
- ▶ Thread depth: Thread length = Distance between workpiece surface and thread termination: Q201
- ▶ Thread pitch: Q239
  The algebraic sign determines the direction of the thread:
  - Right-hand thread: +
  - Left-hand thread: –
- ▶ Coordinate of top of workpiece: Q203
- ▶ 2nd set-up clearance: Q204
- ▶ Infeed depth for chip breaking: Q257
- ▶ Retraction distance for chip breaking: Q256
- ▶ Angle for spindle orientation: Q336

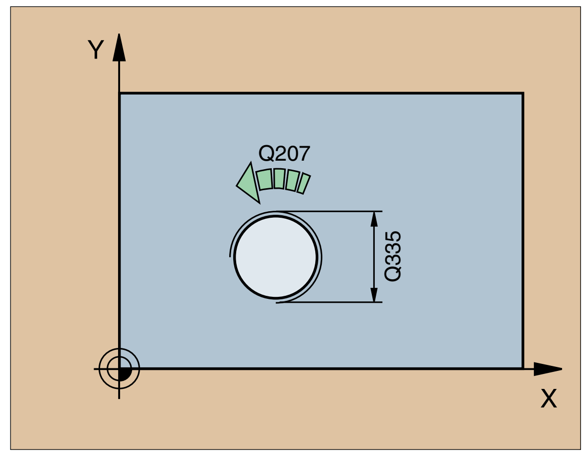
THREAD MILLING (262)

- Pre-position above the hole center with R0
- CYCL DEF: Select Cycle 262 THREAD MILLING
  - Nominal diameter of the thread: Q335
  - Thread pitch: Q239

    The algebraic sign determines the thread direction:
    - Right-hand thread: +
    - Left-hand thread: –
  - Thread depth: Distance from top of workpiece to thread termination: Q201
  - Number of threads per step: Q355
  - Feed rate for pre-positioning: Q253
  - Type of milling: Q351
    - Climb: +1
    - Up-cut: –1
  - Set-up clearance: Q200
  - Workpiece surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Feed rate for milling: Q207

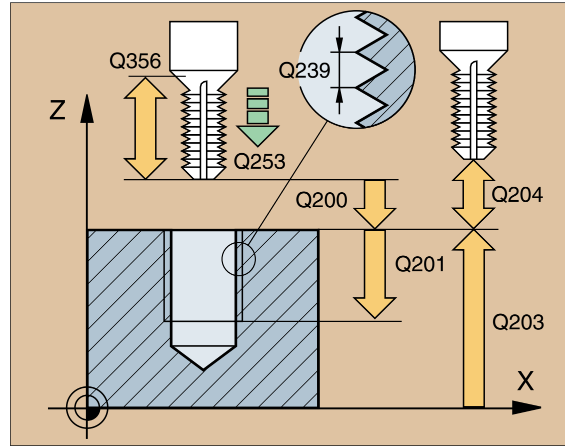
Please note that the TNC executes a compensating movement in the tool axis before approaching. The length of the compensating movement depends on the thread pitch. Make sure that the hole provides sufficient space.

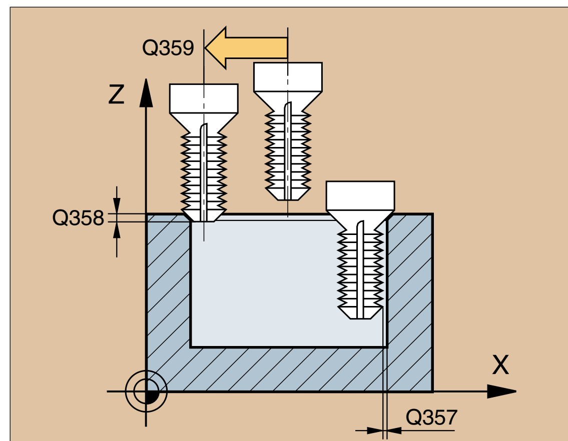




THREAD MILLING AND COUNTERSINKING (263)

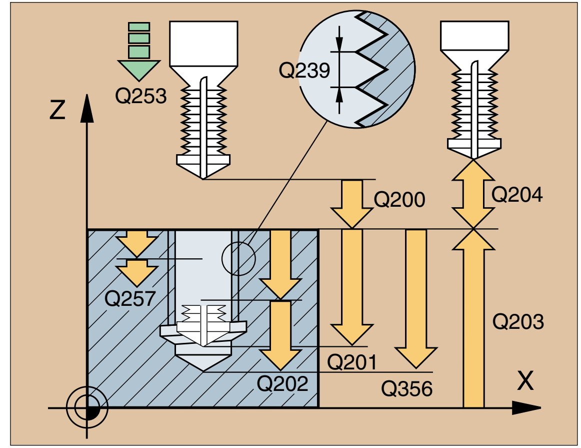
- Pre-position above the hole center with R0
- CYCL DEF: Select Cycle 263 THREAD MILLING AND COUNTERSINKING
  - Nominal diameter of thread: Q335
  - Thread pitch: Q239
    The algebraic sign determines the direction of the thread:
    - Right-hand thread: +
    - Left-hand thread: –
  - Thread depth: Distance from top of workpiece to thread termination: Q201
  - Countersinking depth: Distance from workpiece surface to bottom of hole: Q356
  - Feed rate for pre-positioning: Q253
  - Type of milling: Q351
    - Climb: +1
    - Up-cut: –1
  - Set-up clearance: Q200
  - Lateral set-up clearance: Q357
  - Sinking depth at front: Q358
  - Countersinking offset at front: Q359
  - Workpiece surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Feed rate for counterboring: Q254
  - Feed rate for milling: Q207

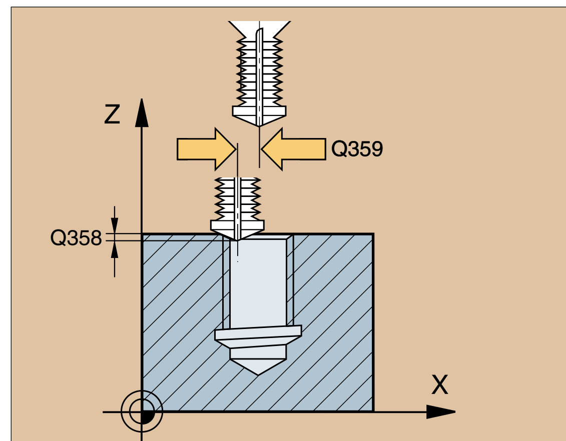




THREAD DRILLING AND MILLING (264)

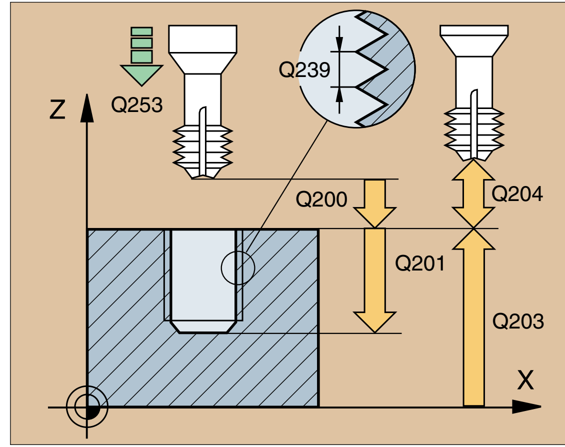
- Pre-position over the hole center with R0
- CYCL DEF: Select Cycle 264 THREAD DRLLNG/MLLNG
  - Nominal diameter of thread: Q335
  - Thread pitch: Q239
    The algebraic sign determines the thread direction:
    - Right-hand thread: +
    - Left-hand thread: –
  - Thread depth: Distance from top of workpiece to thread termination: Q201
  - Hole depth: Distance from top of workpiece to bottom of hole: Q201
  - Feed rate for pre-positioning: Q253
  - Type of milling: Q351
    - Climb: +1
    - Up-cut: –1
  - Plunging depth: Q202
  - Upper advanced stop distance: Q258
  - Infeed depth for chip breaking: Q257
  - Retraction distance for chip breaking: Q256
  - Dwell time at bottom: Q211
  - Sinking depth at front: Q358
  - Countersinking offset at front: Q359
  - Set-up clearance: Q200
  - Workpiece surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Feed rate for plunging: Q206
  - Feed rate for milling: Q207

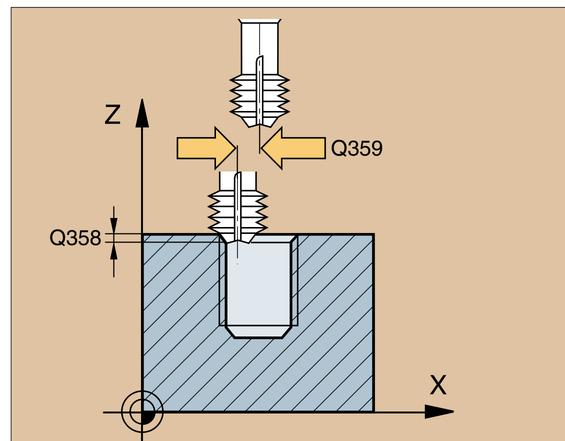




HELICAL THREAD DRILLING AND MILLING (265)

- Pre-position over the hole center with R0
- CYCL DEF: Select Cycle 265 HEL. THREAD DRLG/MLG
  - Nominal diameter of the thread: Q335
  - Thread pitch: Q239
    The algebraic sign determines the thread direction:
    - Right-hand thread: +
    - Left-hand thread: –
  - Thread depth: Distance from top of workpiece to thread termination: Q201
  - Feed rate for pre-positioning: Q253
  - Sinking depth at front: Q358
  - Countersinking offset at front: Q359
  - Countersink: Q360
  - Set-up clearance: Q200
  - Workpiece surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Feed rate for countersinking: Q254
  - Feed rate for milling: Q207

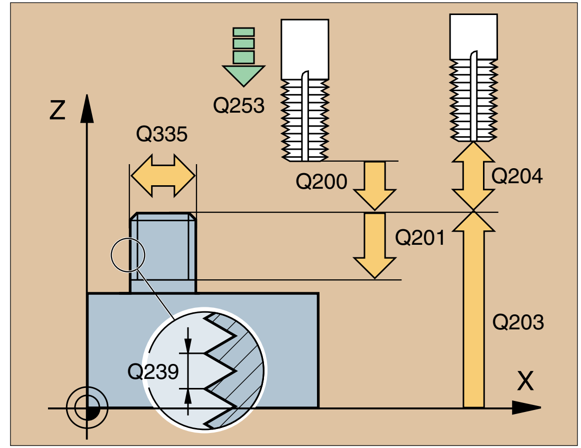


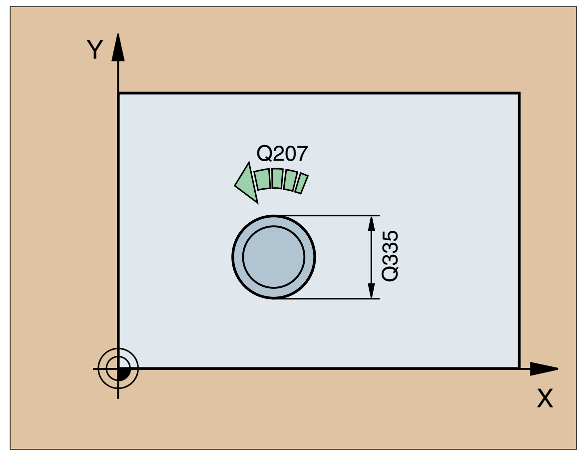


OUTSIDE THREAD MILLING (267)

- Pre-position over the hole center with R0
- CYCL DEF: Select Cycle 267 OUTSIDE THREAD MLLNG
  - Nominal diameter of thread: Q335
  - Thread pitch: Q239

    The algebraic sign determines the thread direction:
    - Right-hand thread: +
    - Left-hand thread: –
  - Hole depth: Distance from top of workpiece to bottom of hole: Q201
  - Number of threads per step: Q355
  - Feed rate for pre-positioning: Q253
  - Type of milling: Q351
    - Climb: +1
    - Up-cut: –1
  - Set-up clearance: Q200
  - Sinking depth at front: Q358
  - Countersinking offset at front: Q359
  - Workpiece surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Feed rate for countersinking: Q254
  - Feed rate for milling: Q207





Pockets, Studs, and Slots

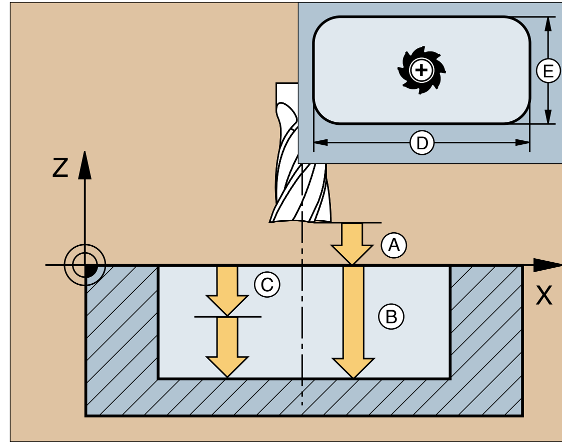
POCKET MILLING (4)

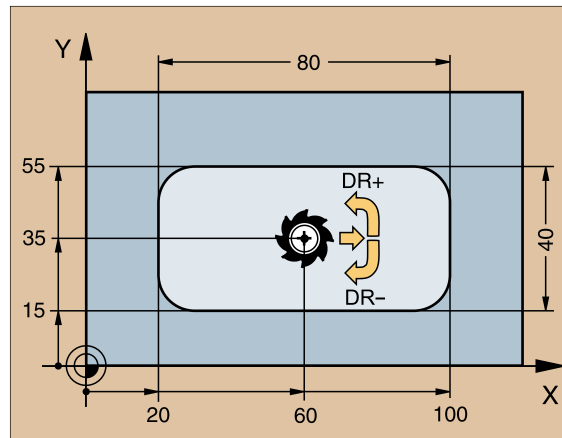
> ⚠ This cycle requires either a center-cut end mill (ISO 1641) or pilot drilling at the pocket center!

The tool begins milling in the positive axis direction of the longer side. In square pockets it moves in the positive Y direction.

- The tool must be pre-positioned over the center of the slot with tool radius compensation **R0**
- CYCL DEF: Select cycle 4 POCKET MILLING
  - Set-up clearance: Ⓐ
  - Milling depth (depth of the pocket): Ⓑ
  - Pecking depth: Ⓒ
  - Feed rate for pecking
  - First side length (length of the pocket, parallel to the first main axis of the working plane): Ⓓ
  - Second side length (width of pocket, sign always positive): Ⓔ
  - Feed rate
  - Rotation clockwise: DR–
    Climb milling with M3: DR+
    Up-cut milling with M3: DR–
  - Rounding-off radius R (radius for the pocket corners)

```
12 CYCL DEF 4.0 POCKET MILLING
13 CYCL DEF 4.1 SET UP2
14 CYCL DEF 4.2 DEPTH-10
15 CYCL DEF 4.3 PECKG4 F80
16 CYCL DEF 4.4 X80
17 CYCL DEF 4.5 Y40
18 CYCL DEF 4.6 F100 DR+ RADIUS 10
19 L Z+100 R0 FMAX M6
20 L X+60 Y+35 FMAX M3
21 L Z+2 FMAX M99
```

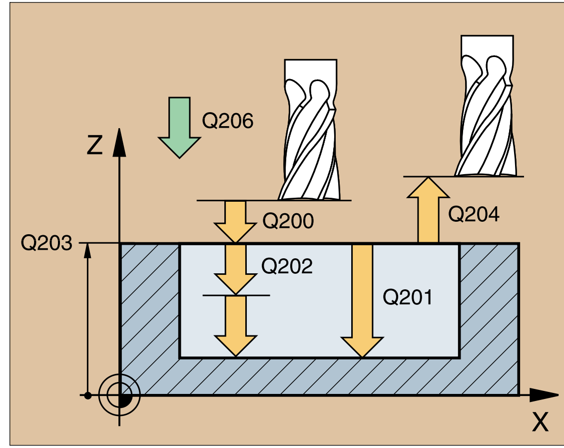


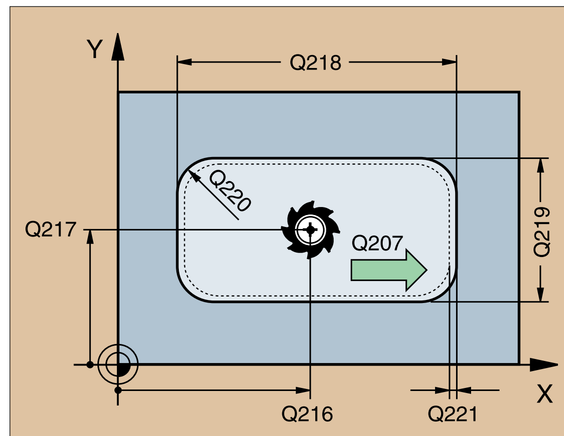


POCKET FINISHING (212)

- CYCL DEF: Select Cycle 212 POCKET FINISHING
  - Set-up clearance: Q200
  - Depth – Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Pecking depth: Q202
  - Feed rate for milling: Q207
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Center in 1st axis: Q216
  - Center in 2nd axis: Q217
  - First side length: Q218
  - Second side length: Q219
  - Corner radius: Q220
  - Allowance in 1st axs: Q221

The TNC automatically pre-positions the tool in the tool axis and in the working plane. If the pecking depth is greater than or equal to the depth, the tool drills to the depth in one plunge

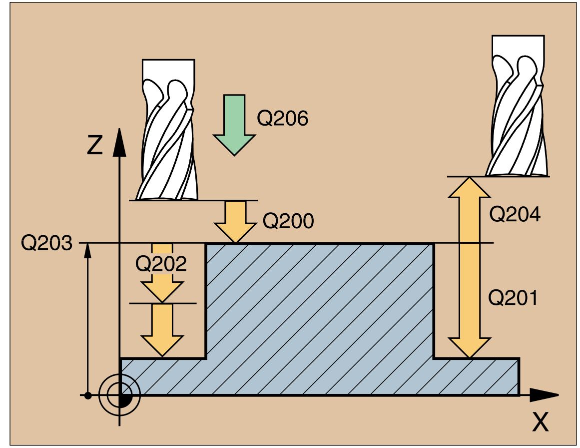


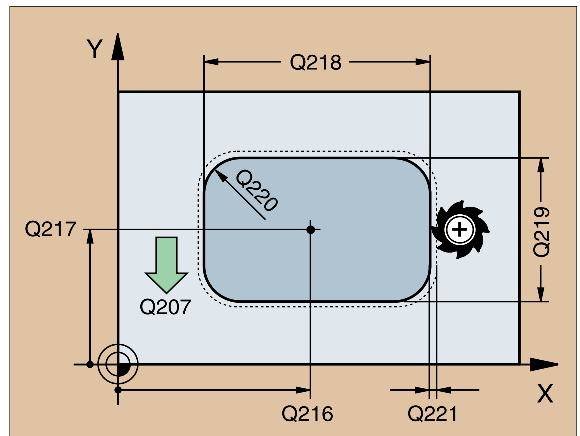


STUD FINISHING (213)

- CYCL DEF: Select Cycle 213 STUD FINISHING
  - Set-up clearance: Q200
  - Depth – Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Pecking depth: Q202
  - Feed rate for milling: Q207
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Center in 1st axis: Q216
  - Center in 2nd axis: Q217
  - First side length: Q218
  - Second side length: Q219
  - Corner radius: Q220
  - Allowance in 1st axs: Q221

The TNC automatically pre-positions the tool in the tool axis and in the working plane. If the pecking depth is greater than or equal to the depth, the tool drills to the depth in one plunge



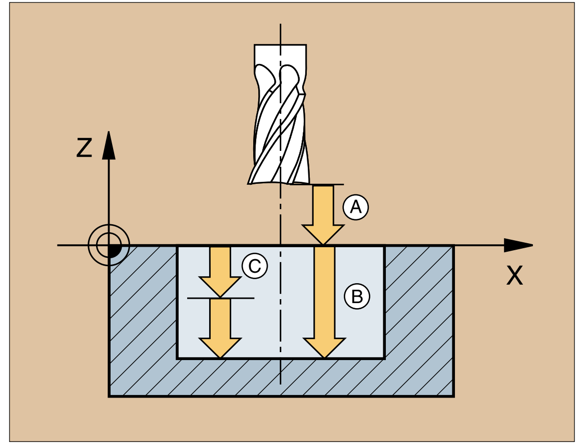


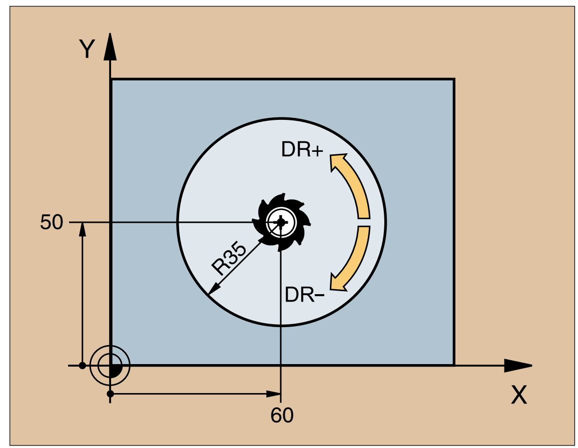
CIRCULAR POCKET MILLING (5)

This cycle requires either a center-cut end mill (ISO 1641) or pilot drilling at pocket center!

- The tool must be pre-positioned over the center of the slot with tool radius compensation **R0**
- CYCL DEF: Select cycle 5
  - Set-up clearance: Ⓐ
  - Milling depth (depth of the pocket): Ⓑ
  - Pecking depth: Ⓒ
  - Feed rate for pecking
  - Circle radius R (radius of the pocket)
  - Feed rate
  - Rotation clockwise: DR–
    Climb milling with M3: DR+
    Up-cut milling with M3: DR–

```
17 CYCL DEF 5.0 CIRCULAR POCKET
18 CYCL DEF 5.1 SET UP 2
19 CYCL DEF 5.2 DEPTH -12
20 CYCL DEF 5.3 PECKG 6 F80
21 CYCL DEF 5.4 RADIUS 35
22 CYCL DEF 5.5 F100 DR+
23 L Z+100 R0 FMAX M6
24 L X+60 Y+50 FMAX M3
25 L Z+2 FMAX M99
```

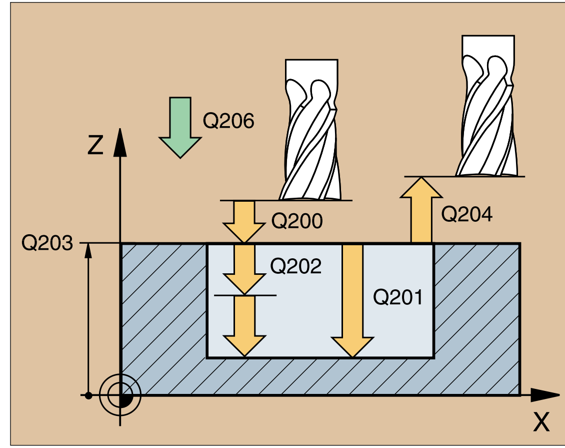


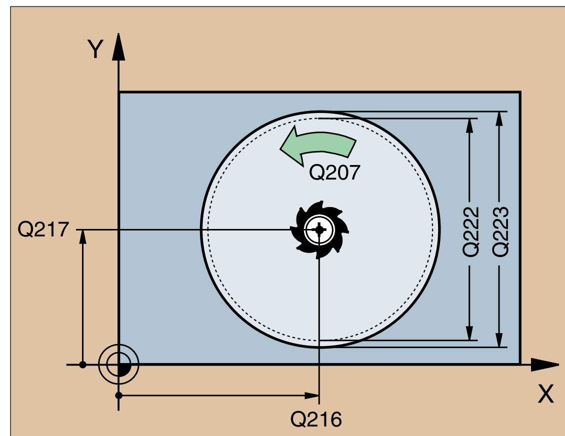


CIRCULAR POCKET FINISHING (214)

- CYCL DEF: Select Cycle 214 CIRCULAR POCKET FINISHING
  - Set-up clearance: Q200
  - Depth – Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Pecking depth: Q202
  - Feed rate for milling: Q207
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Center in 1st axis: Q216
  - Center in 2nd axis: Q217
  - Workpiece blank dia.: Q222
  - Finished part dia.: Q223

The TNC automatically pre-positions the tool in the tool axis and in the working plane. If the pecking depth is greater than or equal to the depth, the tool drills to the depth in one plunge

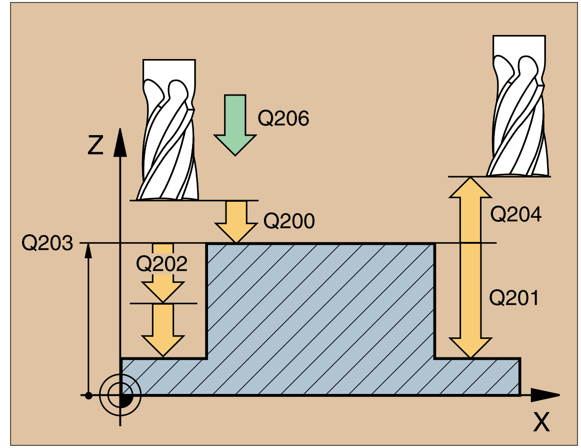


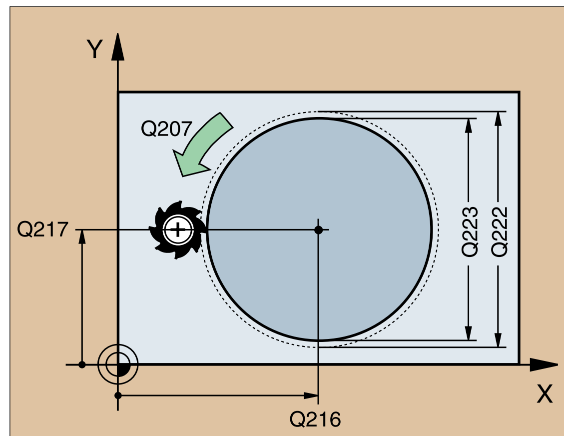


CIRCULAR STUD FINISHING (215)

- CYCL DEF: Select Cycle 215 CIRCULAR STUD FINISHING
  - Set-up clearance: Q200
  - Depth – Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Pecking depth: Q202
  - Feed rate for milling: Q207
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Center in 1st axis: Q216
  - Center in 2nd axis: Q217
  - Workpiece blank dia.: Q222
  - Finished part dia.: Q223

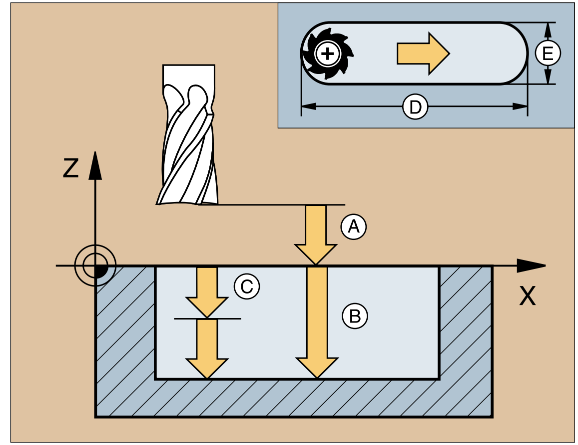
The TNC automatically pre-positions the tool in the tool axis and in the working plane. If the pecking depth is greater than or equal to the depth, the tool drills to the depth in one plunge





SLOT MILLING (3)

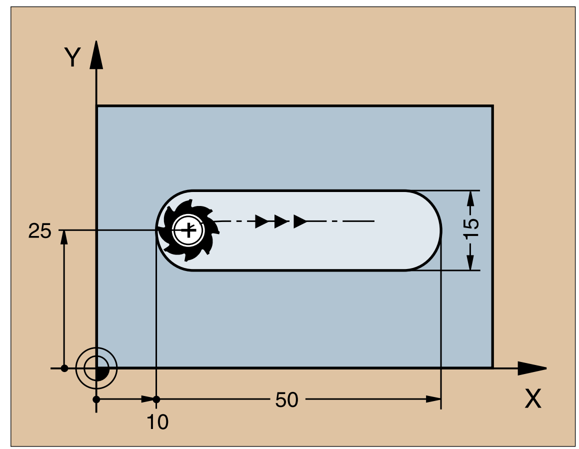
> ⚠
> - This cycle requires either a center-cut end mill (ISO 1641) or pilot drilling at the starting point!
> - The cutter diameter must be smaller than the slot width and larger than half the slot width!



▶ The tool must be pre-positioned over the midpoint of the slot and offset by the tool radius with tool radius compensation at **R0**

▶ CYCL DEF: Select cycle 3 SLOT MILLING
- ▶ Set-up clearance: Ⓐ
- ▶ Milling depth (depth of the slot): Ⓑ
- ▶ Pecking depth: Ⓒ
- ▶ Feed rate for pecking (traverse velocity for plunging)
- ▶ First side length ? (length of the slot): Ⓓ
  The algebraic sign determines the first cutting direction
- ▶ Second side length ? (width of the slot): Ⓔ
- ▶ Feed rate (for milling)

```
10 TOOL DEF 1 L+0 R+6
11 TOOL CALL 1 Z S1500
12 CYCL DEF 3.0 SLOT MILLING
13 CYCL DEF 3.1 SET UP 2
14 CYCL DEF 3.2 DEPTH -15
15 CYCL DEF 3.3 PECKG 5 F80
16 CYCL DEF 3.4 X50
17 CYCL DEF 3.5 Y15
18 CYCL DEF 3.6 F120
19 L Z+100 R0 FMAX M6
20 L X+16 Y+25 R0 FMAX M3
21 L Z+2 M99
```

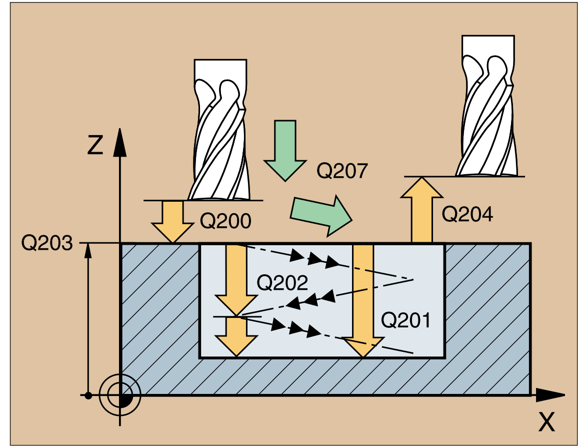


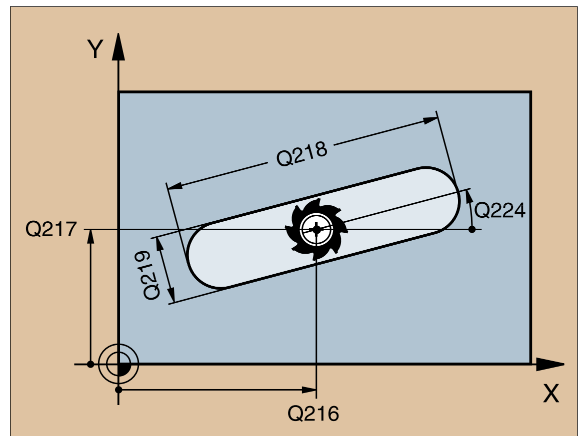
SLOT WITH RECIPROCATING PLUNGE-CUT (210)

> ⚠ The cutter diameter must be no larger than the width of the slot, and no smaller than one third!

- CYCL DEF: Select Cycle 210 SLOT RECIP. PLNG
  - Set-up clearance: Q200
  - Depth – Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for milling: Q207
  - Pecking depth: Q202
  - Machining operation (0/1/2) – 0 = roughing and finishing, 1 = roughing only, 2 = finishing only: Q215
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Center in 1st axis: Q216
  - Center in 2nd axis: Q217
  - First side length: Q218
  - Second side length: Q219
  - Angle of rotation (angle by with the slot is rotated): Q224
  - Infeed for finishing: Q338

The TNC automatically pre-positions the tool in the tool axis and in the working plane. During roughing the tool plunges obliquely into the metal in a back-and-forth motion between the ends of the slot. Pilot drilling is therefore unnecessary.



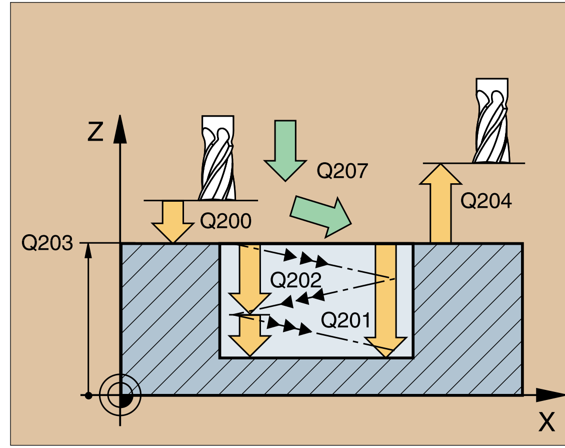


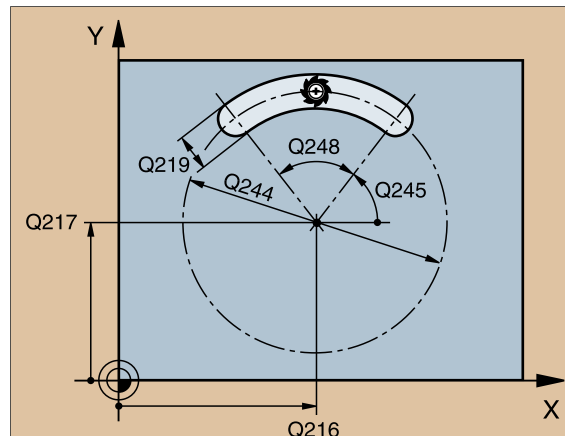
CIRCULAR SLOT with reciprocating plunge (211)

> ⚠ The cutter diameter must be no larger than the width of the slot, and no smaller than one third!

- CYCL DEF: Select Cycle 211 CIRCULAR SLOT
  - Set-up clearance: Q200
  - Depth – Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for milling: Q207
  - Pecking depth: Q202
  - Machining operation (0/1/2) – 0 = roughing and finishing, 1 = roughing only, 2 = finishing only: Q215
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Center in 1st axis: Q216
  - Center in 2nd axis: Q217
  - Pitch circular dia.: Q244
  - Second side length: Q219
  - Starting angle of the slot: Q245
  - Angular length of the slot: Q248
  - Infeed for finishing: Q338

The TNC automatically pre-positions the tool in the tool axis and in the working plane. During roughing the tool plunges obliquely into the metal in a back-and-forth helical motion between the ends of the slot. Pilot drilling is therefore unnecessary.

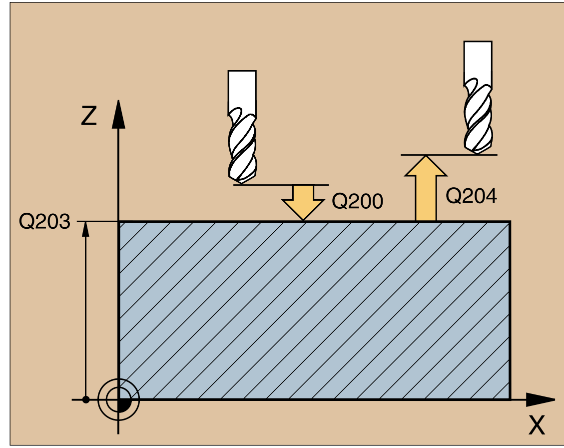




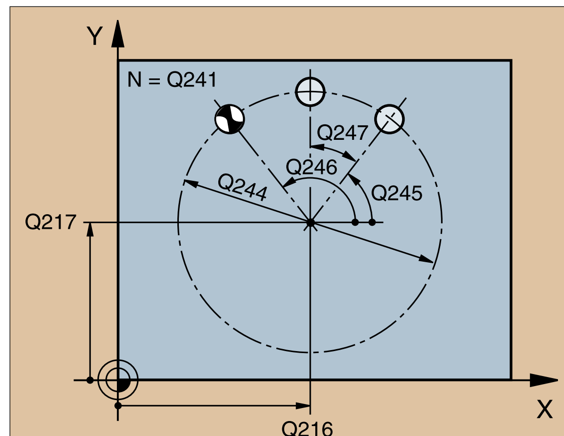
Point Patterns

CIRCULAR PATTERN (220)

- CYCL DEF: Select Cycle 220 CIRCULAR PATTERN
  - Center in 1st axis: Q216
  - Center in 2nd axis: Q217
  - Angle of rotation: Q244
  - Starting angle: Q245
  - Stopping angle: Q246
  - Stepping angle: Q247
  - Nr of repetitions: Q241
  - Set-up clearance: Q200
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Move to clearance height: Q301



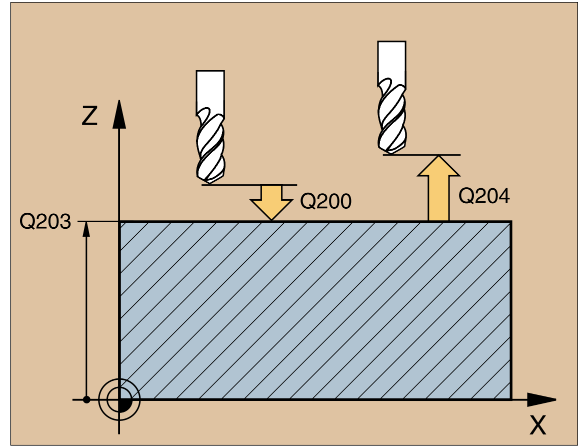
- Cycle 220 POLAR PATTERN is effective immediately upon definition!
- Cycle 220 automatically calls the last defined fixed cycle!
- Cycle 220 can be combined with Cycles 1, 2, 3, 4, 5, 17, 200, 201, 202, 203, 204, 205, 206, 207, 208, 209, 212, 213, 214, 215, 262, 263, 264, 265, 267
- In combined cycles, the set-up clearance, surface coordinate and 2nd set-up-clearance are always taken from Cycle 220!

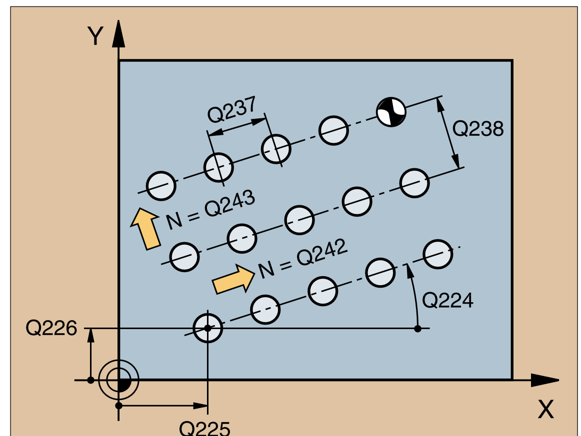


The TNC automatically pre-positions the tool in the tool axis and in the working plane.

LINEAR PATTERN (221)

- CYCL DEF: Select Cycle 221 LINEAR PATTERN
  - Startng pnt 1st axis: Q225
  - Startng pnt 2nd axis: Q226
  - Spacing in 1st axis: Q237
  - Spacing in 2nd axis: Q238
  - Number of columns: Q242
  - Number of lines: Q243
  - Angle of rotation: Q224
  - Set-up clearance: Q200
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Move to clearance height: Q301

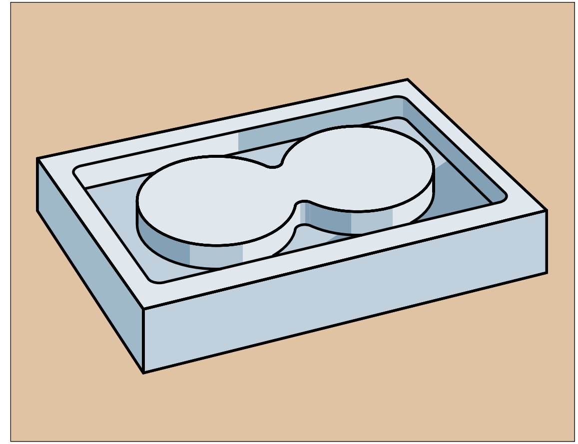




- Cycle 221 LINEAR PATTERN is effective immediately upon definition!
- Cycle 221 automatically calls the last defined fixed cycle!
- Cycle 221 can be combined with Cycles 1, 2, 3, 4, 5, 17, 200, 201, 202, 203, 204, 205, 206, 207, 208, 209, 212, 213, 214, 215, 262, 263, 264, 265, 267
- In combined cycles, the set-up clearance, surface coordinate and 2nd set-up-clearance are always taken from Cycle 221!

The TNC automatically pre-positions the tool in the tool axis and in the working plane.

SL Cycles



General Information

SL cycles are useful when you wish to machine a contour consisting of several subcontours (up to 12 islands or pockets).

The subcontours are defined in subprograms.

When working with subcontours, always remember:

- For a **pocket** the tool machines an inside contour, for an **island** it is an outside contour!
- Tool **approach** and **departure** as well as **infeed in the tool axis cannot** be programmed in SL cycles!
- Each contour listed in Cycle 14 CONTOUR GEOMETRY must be a closed contour!
- There is a limit to the amount of memory an SL cycle can occupy! Approx. 1024 straight line blocks, for example, can be programmed in an SL cycle.

The contour for cycle 25 CONTOUR TRAIN must not be closed!

Make a graphic test run before actually machining a part. That way you can be sure that you defined the contour correctly!

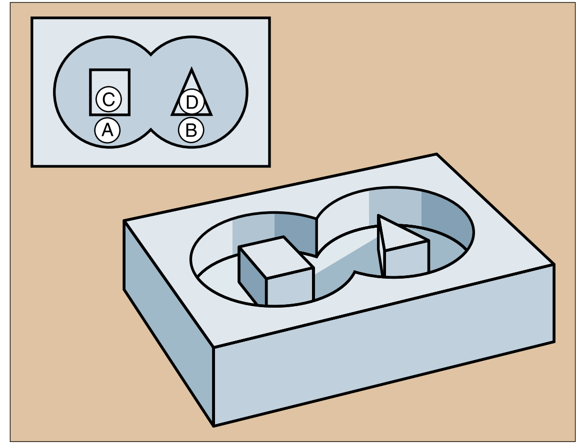
CONTOUR GEOMETRY (14)

In Cycle 14 CONTOUR GEOMETRY you list the subprograms that you wish to superimpose to make a complete closed contour.

- CYCL DEF: Select Cycle 14 CONTOUR GEOMETRY
  - Label nubers for contour: List the LABEL numbers of the subprograms that you wish to superimpose to make a complete closed contour.

Cycle 14 CONTOUR GEOMETRY is effective immediately upon definition!

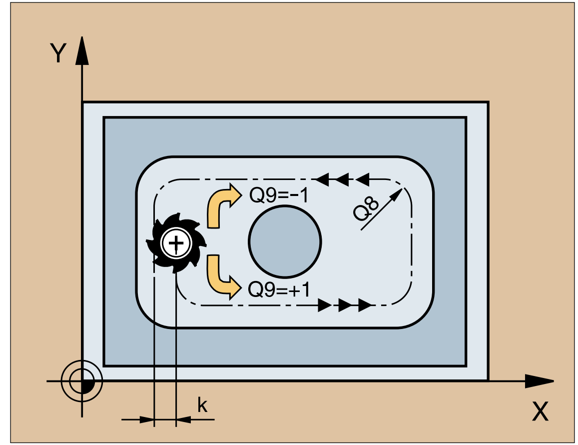
```
4 CYCL DEF 14.0 CONTOUR GEOM
5 CYCL DEF 14.1 CONTOUR LABEL 1/2/3
...
36 L Z+200 R0 FMAX M2
37 LBL1
38 L X+0 Y+10 RR
39 L X+20 Y+10
40 CC X+50 Y+50
...
45 LBL0
46 LBL2
...
58 LBL0
```



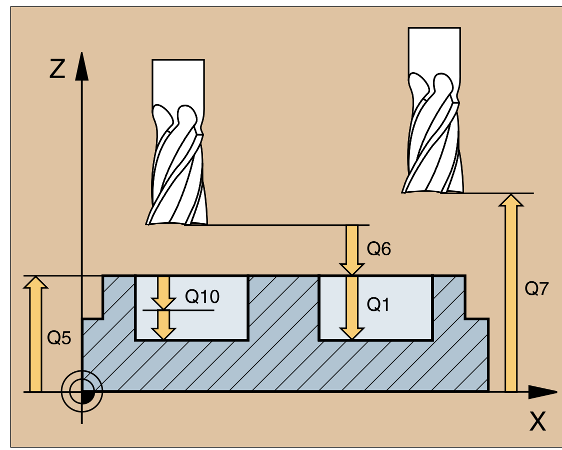
Ⓐ and Ⓑ are pockets, Ⓒ and Ⓓ islands

CONTOUR DATA (20)

Cycle 20 CONTOUR DATA defines the machining information for cycles 21 to 24.



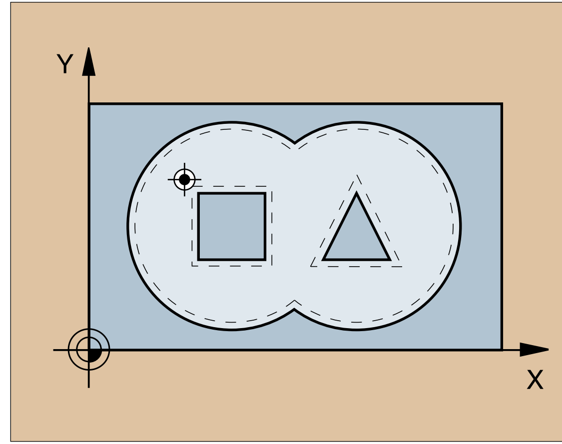
- CYCL DEF: Select Cycle 20 CONTOUR DATA
  - Milling depth Q1:
    Distance from workpiece surface to pocket floor; incremental
  - Path overlap factor Q2:
    Q2 x tool radius = stepover factor k
  - Allowance for side Q3:
    Finishing allowance for the walls of the pocket or island
  - Allowance for floor Q4:
    Finishing allowance for the pocket floor
  - Workpiece surface coordinates Q5:
    Coordinate of the workpiece surface referenced to the current datum; absolute
  - Set-up clearance Q6:
    Distance from the tool to the workpiece surface; incremental
  - Clearance height Q7:
    Height at which the tool cannot collide with the workpiece; absolute
  - Rounding radius Q8:
    Rounding radius of the tool at inside corners
  - Direction of rotation Q9:
    - Clockwise Q9 = –1
    - Counter clockwise Q9 = +1



Cycle 20 CONTOUR DATA is effective immediately upon definition!

PILOT DRILLING (21)

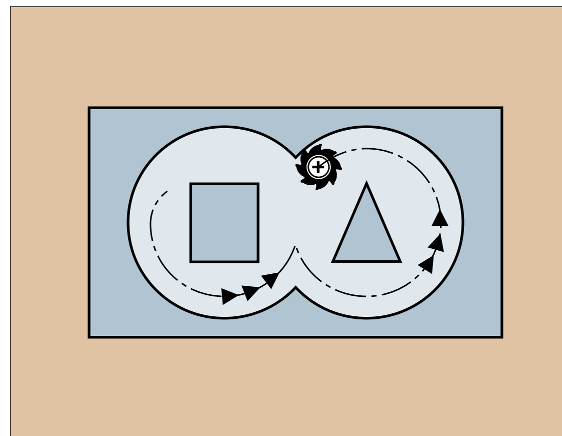
- CYCL DEF: Select Cycle 21 PILOT DRILLING
  - Pecking depth Q10; incremental
  - Feed rate for pecking Q11
  - Rough mill Q13: Number of the roughing tool



ROUGH-OUT (22)

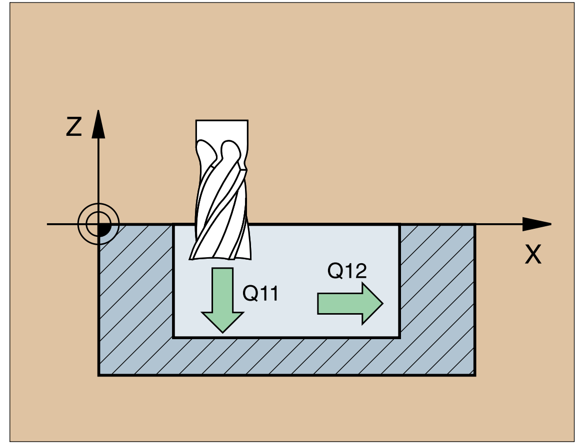
The tool moves parallel to the contour at every pecking depth.

- CYCL DEF: Select Cycle 22 ROUGH-OUT
  - Pecking depth Q10; incremental
  - Feed rate for pecking Q11
  - Feed rate for milling Q12
  - Coarse roughing tool number Q18
  - Feed rate for reciprocation Q19



FLOOR FINISHING (23)

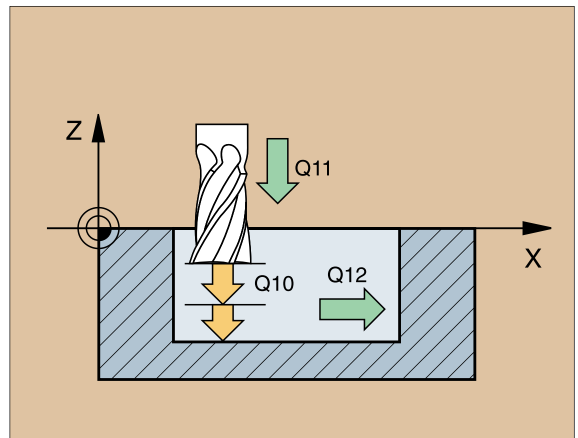
During finishing, the surface is machined parallel to the contour and to the depth previously entered under ALLOWANCE FOR FLOOR.



- CYCL DEF: Select Cycle 23 FLOOR FINISHING
  - Feed rate for pecking Q11
  - Feed rate for milling Q12

SIDE FINISHING (24)

Finishing the individual contour elements



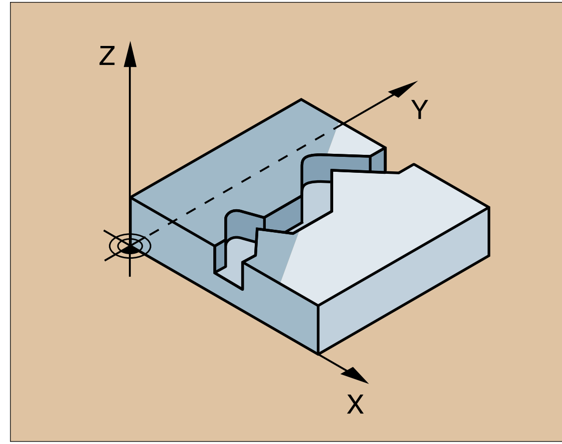
- CYCL DEF: Select Cycle 24 SIDE FINISHING
  - Direction of rotation? Clockwise = –1 Q9:
    - Clockwise Q9 = –1
    - Counterclockwise Q9 = +1
  - Pecking depth Q10; incremental
  - Feed rate for pecking Q11
  - Feed rate for milling Q12
  - Finishing allowance for side Q14: Allowance for finishing in several passes

> - The sum of Q14 + finishing mill radius must be smaller than the sums Q3 (Cycle 20) + roughing tool radius!
> - Call Cycle 22 ROUGH-OUT before calling Cycle 24!

SL Cycles

CONTOUR TRAIN (25)

This cycle is for entering data for machining an open contour that has been defined in a contour subprogam.



- CYCL DEF: Select Cycle 25 CONTOUR TRAIN
  - Milling depth Q1; incremental
  - Allowance for side Q3:
    Finishing allowance in the working plane
  - Workpiece surface coordinates Q5:
    Coordinates referenced to the workpiece datum; absolute
  - Clearance height Q7:
    Height at which the tool cannot collide with the workpiece; absolute
  - Pecking depth Q10; incremental
  - Feed rate for pecking Q11
  - Feed rate for milling Q12
  - Climb or up-cut ? Up-cut = –1 Q15
    - Climb milling: Q15 = +1
    - Up-cut milling: Q15 = –1
    - Alternately in reciprocating cuts: Q15 = 0

- Cycle 14 CONTOUR can have only one label number.
- A subprogram can hold approx. 1024 line segments.
- Do not program incremental dimensions after calling the cycle: danger of collision.
- After calling the cycle, move to a defined absolute position.

CYLINDER SURFACE (27)

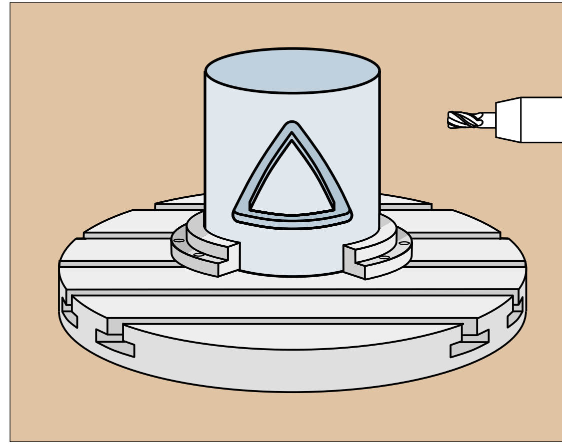
This cycle requires a center-cut end mill (ISO 1641)!

Cycle 27 CYLINDER SURFACE enables you to program a cylindrical contour in only two axes, as if in a plane. The TNC then rolls it onto a cylindrical surface.

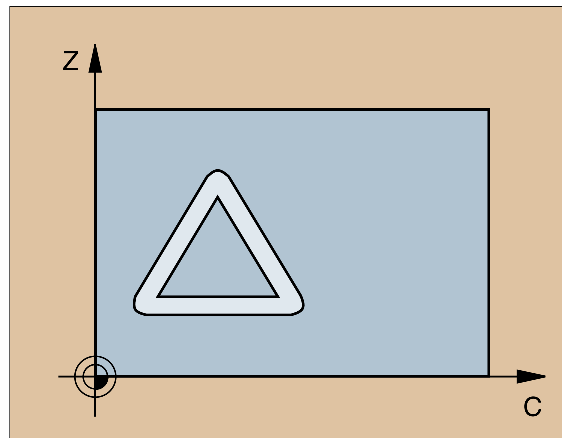
- Define a contour in a subprogram and list it in Cycle 14 CONTOUR GEOMETRY
- CYCL DEF: Select Cycle 27 CYLINDER SURFACE
  - Milling depth Q1
  - Finishing allowance for side Q3: Enter the finishing allowance (Either Q3>0 or Q3<0)
  - Set-up clearance ? Q6: Distance from the tool to the workpiece
  - Plunging depth Q10
  - Feed rate for plunging Q11
  - Feed rate for milling Q12
  - Cylinder radius Q16: Radius of the cylinder
  - Dimension type? Deg=0 mm/inch=1 Q17: You can enter coordinates in the subprogram in degrees or millimeters

The machine and TNC must be prepared for the CYLINDER SURFACE cycle by the machine tool builder!

- The workpiece must be set up concentrically on the rotary table!
- The tool axis must be perpendicular to the axis of the rotary table!
- Cycle 14 CONTOUR GEOMETRY can have only one label number!
- A subprogram can hold approx. 1024 line segments!



▼ The unrolled contour



CYLINDER SURFACE (28)

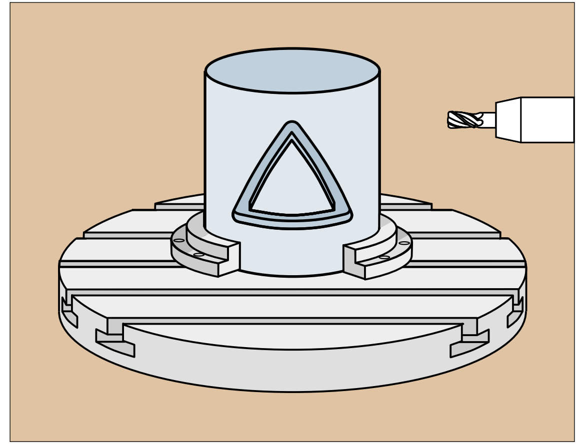
This cycle requires a center-cut end mill (ISO 1641)!

Cycle 28 CYLINDER SURFACE enables you to program a slot in only two axes and then machine it on a cylindrical surface without distorting the angle of the slot walls.

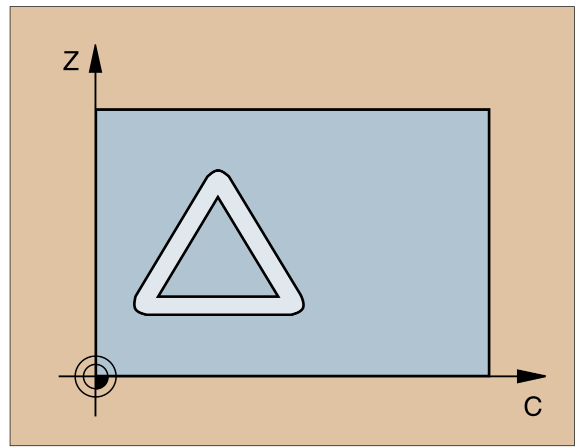
- Define a contour in a subprogram and list it in Cycle 14 CONTOUR GEOMETRY.
- CYCL DEF: Select Cycle 28 CYLINDER SURFACE
  - Milling depth Q1
  - Finishing allowance for side Q3: Enter the finishing allowance (Q3>0 or Q3<0)
  - Set-up clearance Q6: Distance from the tool to the workpiece surface
  - Plunging depth Q10
  - Feed rate for plunging Q11
  - Feed rate for milling Q12
  - Cylinder radius Q16: Radius of the cylinder
  - Dimension type? Deg=0 mm/inch=1 Q17: Coordinates in the subprogram in degrees or millimeters
  - Slot width Q20

- The machine and TNC must be prepared for the CYLINDER SURFACE CYCLE by the machine tool builder!

- The workpiece must be set up concentrically on the table!
- The tool axis must be perpendicular to the rotary table axis!
- Cycle 14 CONTOUR GEOMETRY can have only one label number!
- A subprogram can hold approx. 1024 line segments!



▼ The unrolled contour



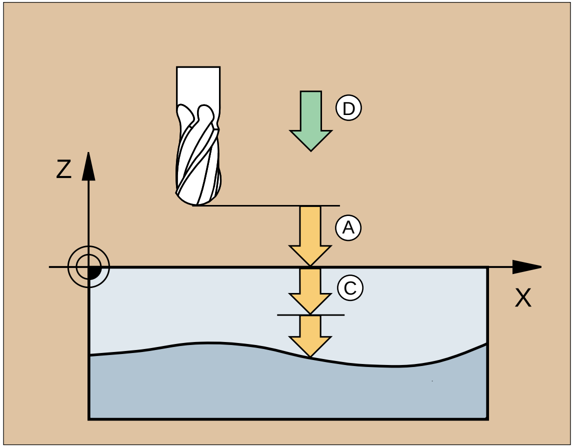
Cycles for Multipass Milling

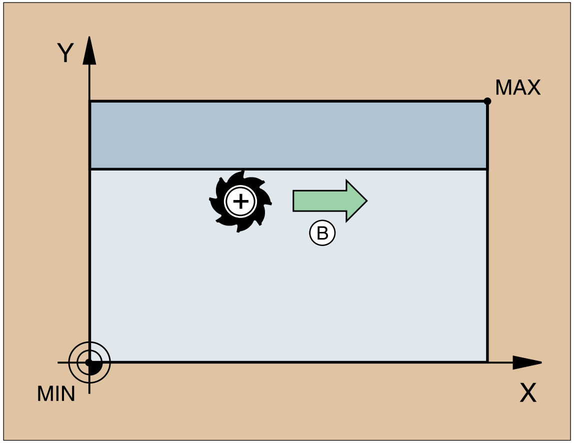
3-D DATA (30)

This cycle requires a center-cut end mill as per ISO 1641!

- CYCL DEF: Select Cycle 30 3-D DATA
  - pgm name for digitized data
  - MIN. point range
  - MAX. point range
  - Set-up clearance: Ⓐ
  - Pecking depth: Ⓒ
  - Feed rate for pecking: Ⓓ
  - Feed rate: Ⓑ
  - Miscellaneous function M

```
7 CYCL DEF 30.0 3-D DATA
8 CYCL DEF 30.1 PROGRAM1
9 CYCL DEF 30.2 X+0 Y+0 Z-35
10 CYCL DEF 30.3 X+250 Y+125 Z+15
11 CYCL DEF 30.4 SET UP 2
12 CYCL DEF 30.5 PECKG 5 F125
13 CYCL DEF 30.6 F350 M112 T0.01 A+10
```

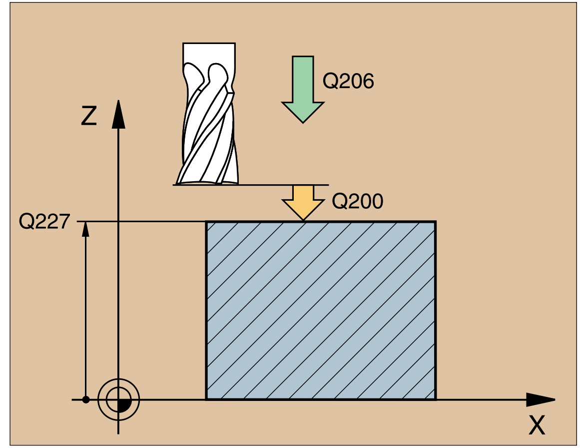


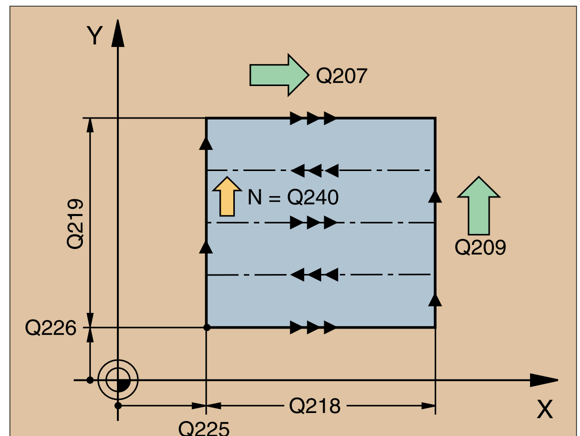


MULTIPASS MILLING (230)

From the current position, the TNC positions the tool automatically at the starting point of the first machining operation, first in the working plane and then in the tool axis. Pre-position the tool in such a way that there is no danger of collision with the workpiece or fixtures.

- CYCL DEF: Select Cycle 230 MULTIPASS MILLING
  - Starting point in 1st axis: Q225
  - Starting point in 2nd axis: Q226
  - Starting point in 3rd axis: Q227
  - First side length: Q218
  - Second side length: Q219
  - Number of cuts: Q240
  - Feed rate for plunging: Q206
  - Feed rate for milling: Q207
  - Stepover feed rate: Q209
  - Set-up clearance: Q200

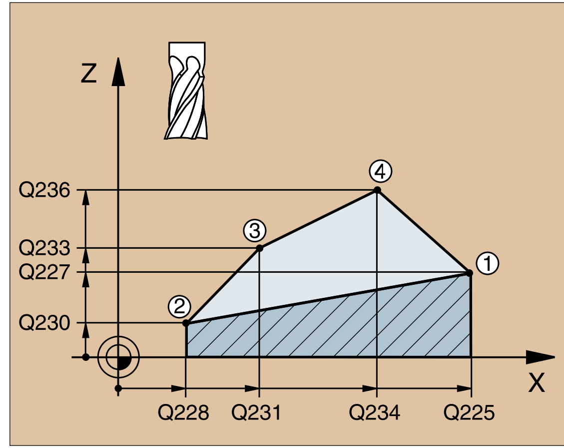


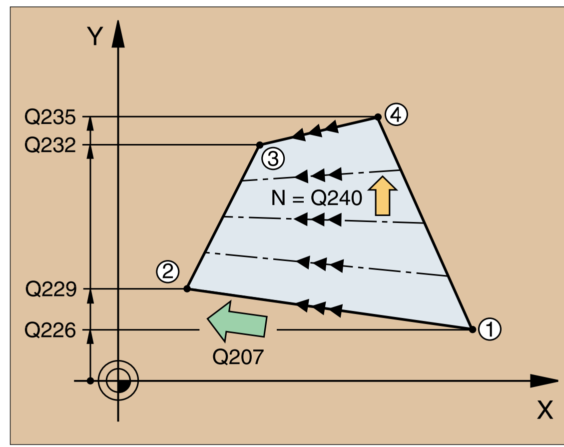


RULED SURFACE (231)

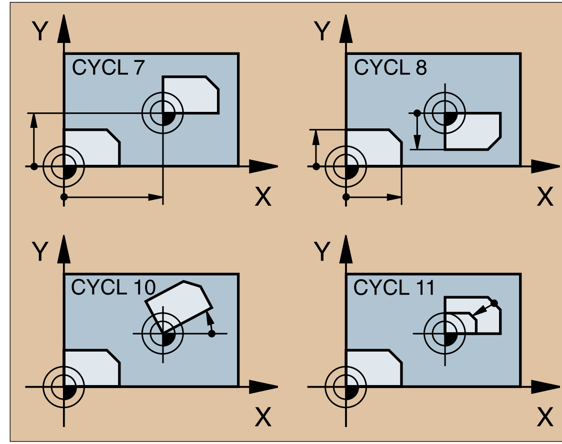
Starting from the initial position, the TNC positions the tool at the starting point (point 1), first in the working plane and then in the tool axis.

- CYCL DEF: Select Cycle 231 RULED SURFACE
  - Starting point in 1st axis: Q225
  - Starting point in 2nd axis: Q226
  - Starting point in 3rd axis: Q227
  - 2nd point in 1st axis: Q228
  - 2nd point in 2nd axis: Q229
  - 2nd point in 3rd axis: Q230
  - 3rd point in 1st axis: Q231
  - 3rd point in 2nd axis: Q232
  - 3rd point in 3rd axis: Q233
  - 4th point in 1st axis: Q234
  - 4th point in 2nd axis: Q235
  - 4th point in 3rd axis: Q236
  - Number of cuts: Q240
  - Feed rate for milling: Q207





Cycles for Coordinate Transformation

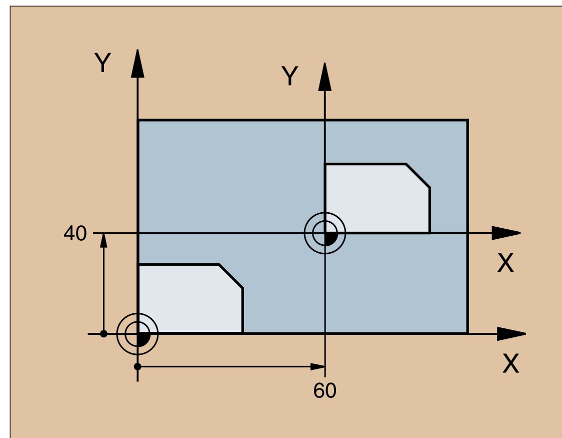


Cycles for coordinate transformation permit contours to be

• Shifted	Cycle	7	DATUM SHIFT
• Mirrored	Cycle	8	MIRROR IMAGE
• Rotated (in the plane)	Cycle	10	ROTATION
• Tilted out of the plane	Cycle	19	WORKING PLANE
• Enlarged or reduced	Cycle	11	SCALING
	Cycle	26	AXIS-SPECIFIC SCALING

Cycles for coordinate transformation are effective upon definition until they are reset or redefined. The original contour should be defined in a subprogram. Input values can be both absolute and incremental.

DATUM SHIFT (7)



- CYCL DEF: Select Cycle 7 DATUM SHIFT
  - Enter the coordinates of the new datum or the number of the datum from the datum table.

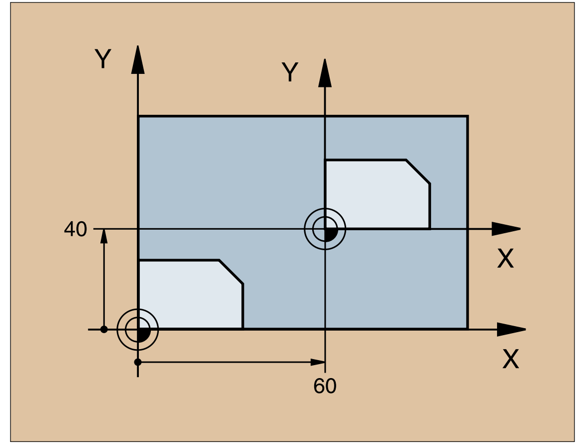
To cancel a datum shift: Re-enter the cycle definition with the input value 0.

9 CALL LBL1	Call the part subprogram
10 CYCL DEF 7.0 DATUM SHIFT	
11 CYCL DEF 7.1 X+60	
12 CYCL DEF 7.2 Y+40	
13 CALL LBL1	Call the part subprogram

When combining transformations, the datum shift must be programmed before the other transformations!

DATUM SETTING (247)

▶ CYCL DEF: Select Cycle 247 DATUM SETTING
  ▶ Datum number: Enter the number from the active datum table containing the REF coordinates of the datum to be set.

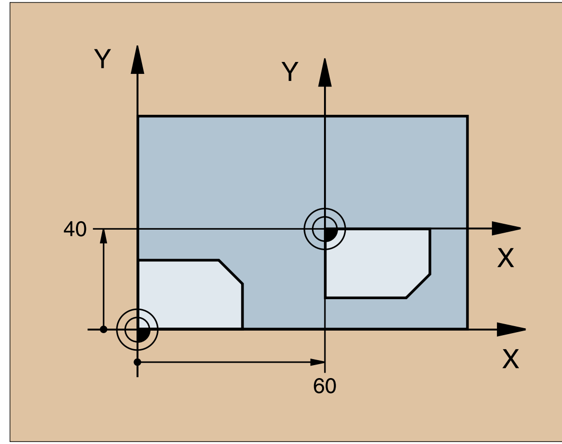


Reset

You can reactivate the datum that was last set in the Manual operating mode by entering the miscellaneous function M104.

- If required, activate the desired datum table with the NC block SEL TABLE.
- The TNC sets the datum only in the axes that are active in the datum table.
- Cycle 247 always interprets the values saved in the datum tables as coordinates relative to the machine datum. Machine parameter 7475 has no influence.

MIRROR IMAGE (8)



- CYCL DEF: Select Cycle 8 MIRROR IMAGE
  - Enter the mirror image axis: Either X, Y, or both

To reset the mirror image, re-enter the cycle definition with NO ENT.

```
15 CALL LBL1
16 CYCL DEF 7.0 DATUM SHIFT
17 CYCL DEF 7.1 X+60
18 CYCL DEF 7.2 Y+40
19 CYCL DEF 8.0 MIRROR IMAGE
20 CYCL DEF 8.1 Y
21 CALL LBL1
```

- The tool axis cannot be mirrored!
- The cycle always mirrors the original contour (in this example in subprogram LBL1)!

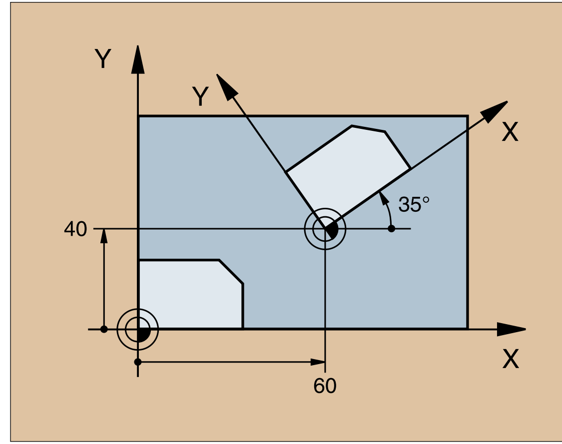
Rotation (10)

- CYCL DEF: Select Cycle 10 ROTATION
  - Enter the rotation angle:
    - Input range –360° to +360°
    - Reference axes for the rotation angle

Working plane	Reference axis and 0° direction
X/Y	X
Y/Z	Y
Z/X	Z

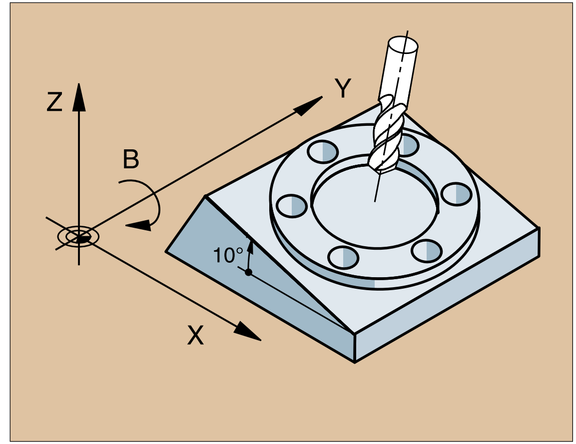
To reset a ROTATION, re-enter the cycle with the rotation angle 0.

```
12 CALL LBL1
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 10.0 ROTATION
17 CYCL DEF 10.1 ROT+35
18 CALL LBL1
```



WORKING PLANE (19)

Cycle 19 WORKING PLANE supports machining operations with a swivel head and/or tilting table.



- Call the tool
- Retract the tool in the tool axis (to prevent collision)
- If required, use an L block to position the rotary axes to the desired angle
- CYCL DEF: Select Cycle 19 WORKING PLANE
  - Enter the tilt angle of the corresponding axis or angle in space
  - If required, enter the feed rate of the rotary axes during automatic positioning
  - If required, enter the setup-clearance
- Activate compensation: move all the axes
- Program the contour as if the plane were not tilted

To cancel the WORKING PLANE cycle, re-enter the cycle definition with a 0° angle.

The machine and TNC must be prepared for the WORKING PLANE cycle by the machine tool builder!

```
4 TOOL CALL 1 Z S2500
5 L Z+350 R0 FMAX
6 L B+10 C+90 R0 FMAX
7 CYCL DEF 19.0 WORKING PLANE
8 CYCL DEF 19.1 B+10 C+90
9 L Z+200 R0 F1000
10 L X-50 Y-50 R0
```

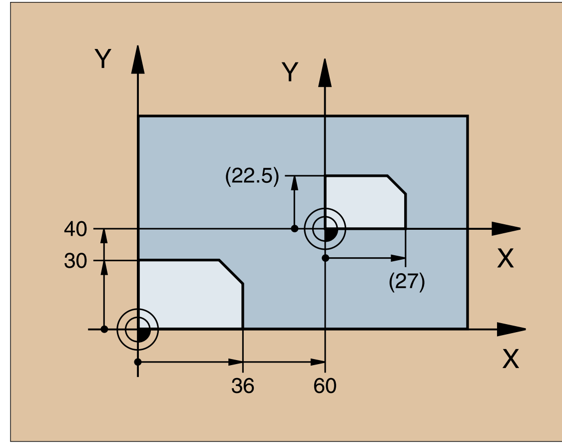
SCALING (11)

▶ CYCL DEF: Select Cycle 11 SCALING
  ▶ Enter the scaling factor (SCL):
    - Input range 0.000001 to 99.999999:
      To reduce the contour ... SCL < 1
      To enlarge the contour ... SCL > 1

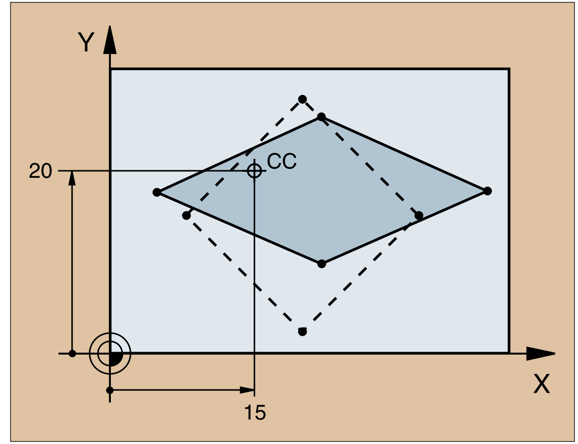
To cancel the SCALING, re-enter the cycle definition with SCL1.

```
11 CALL LBL1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL1
```

SCALING can be effective in the working plane only or in all three main axes (depending on machine parameter 7410)!



AXIS-SPECIFIC SCALING (26)



- CYCL DEF: Select Cycle 20 AXIS-SPEC. SCALING
  - Axis and factor: Coordinate axes and factors for extending or compressing contour dimensions
  - Centerpoint coord. of extention: Center of the extension or compression

To cancel the AXIS-SPEC. SCALING, re-enter the cycle definition assigning the factor 1 to the affected axes.

> Coordinate axes sharing coordinates for arcs must be extended or compressed by the same scaling factor!

```
25 CALL LBL1
26 CYCL DEF 26.0 AXIS-SPEC. SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL1
```

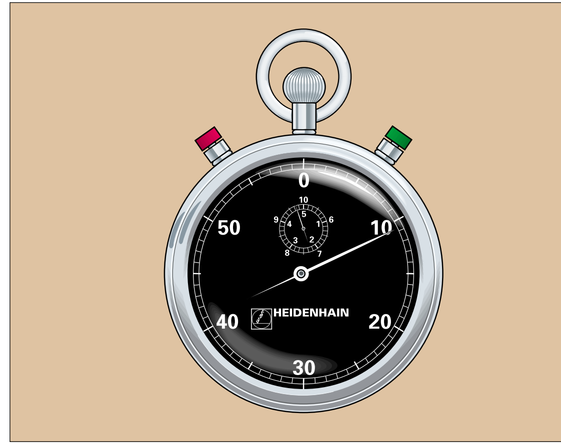
Special Cycles

DWELL TIME (9)

The program run is interrupted for the duration of the DWELL TIME.

▶ CYCL DEF: Select cycle 9 DWELL TIME
  ▶ Enter the dwell time in seconds

```
48 CYCL DEF 9.0 DWELL TIME
49 CYCL DEF 9.1 DWELL 0.5
```

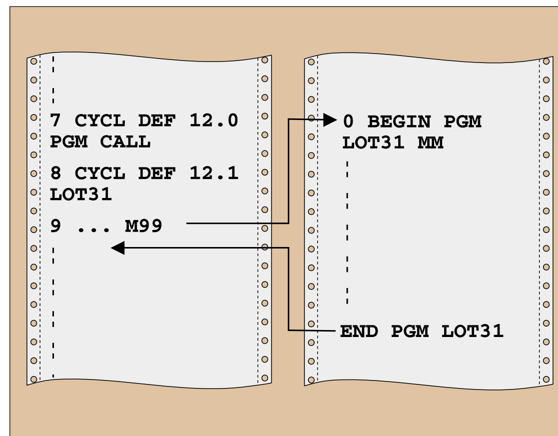


PGM CALL (12)

▶ CYCL DEF: Select cycle 12 PGM CALL
  ▶ Enter the name of the program that you wish to call

Cycle 12 PGM CALL must be called to become active!

```
7 CYCL DEF 12.0 PGM CALL
8 CYCL DEF 12.1 LOT31
9 L X+37.5 Y-12 R0 FMAX M99
```

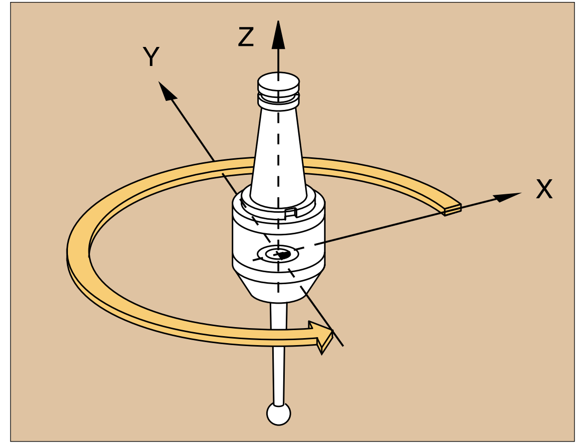


Spindle ORIENTATION

- CYCL DEF: Select cycle 13 ORIENTATION
  - Enter the orientation angle referenced to the angle reference axis of the working plane:
    - Input range 0 to 360°
    - Input resolution 0.1°
- Call the cycle with M19 or M20

The machine and TNC must be prepared for spindle ORIENTATION by the machine tool builder!

```
12 CYCL DEF 13.0 ORIENTATION
13 CYCL DEF 13.1 ANGLE 90
```



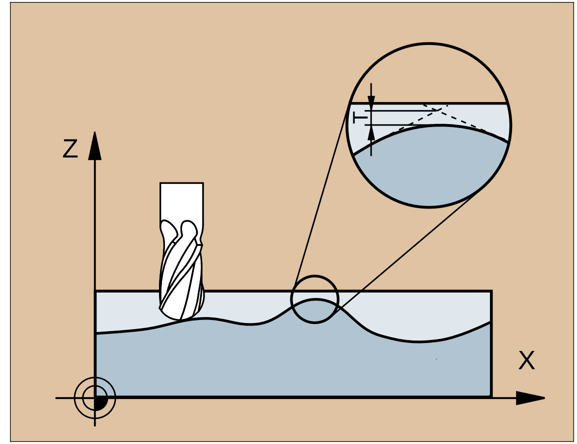
TOLERANCE (32)

The machine and the TNC must be specially prepared for fast contour milling by the machine tool builder!

Cycle 32 TOLERANCE is effective as soon as it is defined in the part program!

The TNC automatically smoothes the contour between any (compensated or uncompensated) contour elements. The tool therefore moves continuously on the workpiece surface. If necessary, the TNC automatically reduces the programmed feed rate so that the program can be run at the **fastest possible** speed and without "jerk".

A contour deviation results from the smoothing out. The size of this deviation (TOLERANCE VALUE) is set in a machine parameter by the machine manufacturer. You can change the pre-set tolerance value with Cycle 32 (see figure at top right).



- CYCL DEF: Select Cycle 32 TOLERANCE
  - Tolerance T: permissible contour deviation in mm
  - Finishing/Roughing: Select the filter settings
    0: Milling with increased contour accuracy
    1: Milling at increased feed rate
  - Tolerance for rotary axes: Permissible position error of rotary axes in degrees with active M128.

Graphics and Status Displays

See "Graphics and Status Displays"

Defining the Workpiece in the Graphic Window

The dialog prompt for the BLK-FORM appears automatically whenever you create a new part program.

- Create a new program or, if you are already in a program, press the soft key BLK FORM
  - Spindle axis
  - MIN and MAX point

The following is a selection of frequently needed functions.

Interactive Programming Graphics

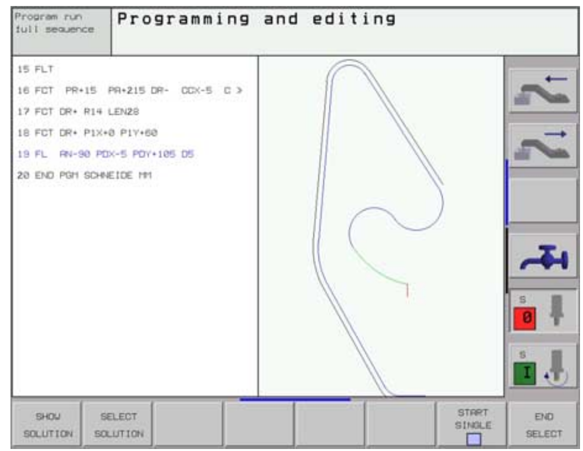
Select the PGM+GRAPHICS screen layout!

The TNC can generate a two-dimensional graphic of the contour while you are programming it:

- Automatic graphic generation during programming

- Manually start graphic generation

- Generate interactive graphics blockwise



Test Graphics and Program Run Graphics

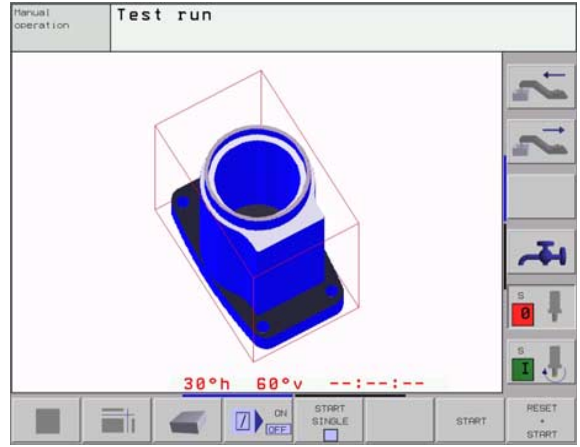
Select the GRAPHICS or PGM+GRAPHICS screen layout!

In the test run and program run modes the TNC can graphically simulate the machining process. The following display types are available via soft key:

- Plan view

- Projection in three planes

- 3D view



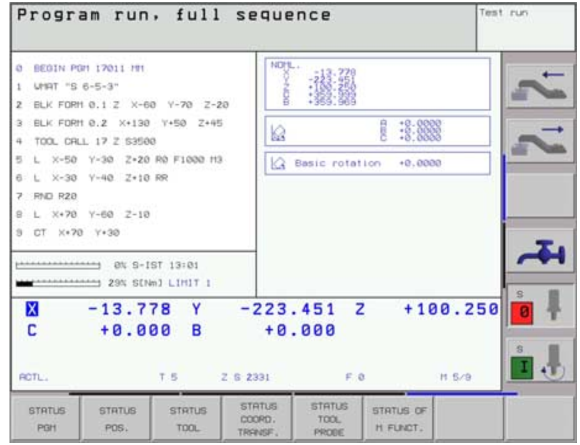
Status Displays

Select the PGM+STATUS or POSITION+STATUS screen layout!

In the program run modes a window in the lower part of the screen shows information on

- Tool position
- Feed rate
- Active miscellaneous functions

Further status information is available via soft key for display in an additional window:



Soft key	Function
STATUS PGM	Program information
STATUS POS.-ANZ.	Tool positions
STATUS WERKZEUG	Tool data
STATUS KOORD. UMRECHN.	Coordinate transformations
STATUS CALL LBL	Subprograms, program section repeat
STATUS TOOL PROBE	Tool measurement
STATUS OF M FUNCT.	Active miscellaneous functions M

ISO Programming

Programming Tool Movements with Cartesian Coordinates

G00	Linear motion in rapid traverse
G01	Linear motion
G02	Circular motion, clockwise
G03	Circular motion, counterclockwise
G05	Circular motion without directional data
G06	Circular movement with tangential contour connection
G07*	Paraxial positioning block

Programming Tool Movements with Polar Coordinates

G10	Linear motion in rapid traverse
G11	Linear motion
G12	Circular motion, clockwise
G13	Circular motion, counterclockwise
G15	Circular motion without directional data
G16	Circular movement with tangential contour connection

Drilling Cycles

G83	Pecking
G200	Drilling
G201	Reaming
G202	Boring
G203	Universal boring
G204	Back boring
G205	Universal pecking
G208	Bore milling
G84	Tapping
G206	Tapping NEW
G85	Rigid tapping (controlled spindle)
G207	Rigid tapping (controlled spindle) NEW
G86	Thread cutting
G209	Tapping with chip breaking
G262	Thread milling
G263	Thread milling and countersinking
G264	Thread drilling and milling
G265	Helical thread drilling and milling
G267	Outside thread milling

*) Effective blockwise

Pockets, Studs and Slots

G75	Rectangular pocket milling, clockwise machining direction
G76	Rectangular pocket milling, counterclockwise machining direction
G212	Pocket milling
G213	Stud milling
G77	Circular pocket milling, clockwise machining direction
G78	Circular pocket milling, counterclockwise machining direction
G214	Circular pocket finishing
G215	Circular stud finishing
G74	Slot milling
G210	Slot milling with reciprocating plunge
G211	Circular slot

Point Patterns

G220	Circular point pattern
G221	Linear point pattern

SL Cycles, Group I

G37	List of contour subprograms
G56	Pilot drilling
G57	Rough-out
G58	Contour milling, clockwise
G59	Contour milling, counterclockwise

SL Cycles, Group II

G37	List of contour subprograms
G120	Contour data
G121	Pilot drilling
G122	Rough-out
G123	Floor finishing
G124	Side finishing
G125	Contour train
G127	Cylinder surface
G128	Cylinder surface slot milling

Multipass milling

G60	3-D data
G230	Multipass milling
G231	Ruled surface

Cycles for Coordinate Transformation

G53	Datum shift from datum tables
G54	Entering datum shift directly
G247	Datum setting
G28	Mirror image
G73	Rotating the coordinate system
G72	Scaling factor: enlarging/reducing contours
G80	Working plane

Special Cycles

G04*	Dwell time
G36	Oriented spindle stop
G39	Designating a program as a cycle
G79*	Cycle call

Touch Probe Cycles

G55*	Measure coordinate
G400*	Basic rotation over 2 points
G401*	Basic rotation over 2 holes
G402*	Basic rotation over 2 studs
G403*	Basic rotation over a rotary table
G404*	Set basic rotation
G405*	Basic rotation over rotary table, hole center

Touch Probe Cycles

G410*	Datum at center of rectangular pocket
G411*	Datum at center of rectangular stud
G412*	Datum at center of hole
G413*	Datum at center of circular stud
G414*	Datum at outside corner
G415*	Datum at inside corner
G416*	Datum at center of bolt hole circle
G417*	Datum in touch probe axis
G418*	Datum at center of 4 holes
G420*	Measure angle
G421*	Measure hole
G422*	Measure circular stud
G423*	Measure rectangular pocket
G424*	Measure rectangular stud
G425*	Measure slot width
G426*	Measure ridge width
G427*	Measure any coordinate
G430*	Measure bolt hole circle
G431*	Measure plane
G440*	Thermal compensation
G480*	Calibrate TT
G481*	Measuring tool length
G482*	Measuring tool length
G483*	Measuring tool length and radius

*) Effective blockwise

Defining the Working Plane

G17	X/Y working plane, tool axis Z
G18	Z/X working plane, tool axis Y
G19	Y/Z working plane, tool axis X
G20	Fourth axis is tool axis

Chamfer, Rounding, Approach/Departure

G24*	Chamfer with side length R
G25*	Corner rounding with radius R
G26*	Tangential contour approach on an arc with radius R
G27*	Tangential contour departure on an arc with radius R

Tool Definition

G99*	Tool definition in the program with length L and radius R

Tool Radius Compensation

G40	No radius compensation
G41	Radius compensation to the left of the contour
G42	Radius compensation to the right of the contour
G43	Paraxial radius compensation: the path is lengthened
G44	Paraxial radius compensation: the path is shortened

Dimensional Data

G90	Absolute dimensions
G91	Incremental (chain) dimensions

Unit of Measure (at Beginning of Program)

G70	Inches
G71	Millimeters

Blank Form Definition for Graphics

G30	Setting the working plane, MIN point coordinates
G31	Dimensional data (with G90, G91), coordinates of the MAX point

Other G functions

G29	Define last nominal position value as pole
G38	Stopping the program run
G51*	Calling the next tool (only with central tool file)
G98*	Setting a label number

*) Effective blockwise

Q Parameter Functions	
D00	Assign a value directly
D01	Calculate and assign the sum of two values
D02	Calculate and assign the difference of two values
D03	Calculate and assign the product of two values
D04	Calculate and assign the quotient of two values
D05	Calculate and assign the root from a value
D06	Calculate and assign the sine of an angle in degrees
D07	Calculate and assign the cosine of an angle in degrees
D08	Calculate and assign the square root of the sum of two squares (Pythagorean theorem)
D09	If equal, jump to the given label
D10	If not equal, jump to the given label
D11	If greater than, jump to the given label
D12	If less than, jump to the given label
D13	Find and assign an angle from the arc tangent of two sides or from the sine and cosine of an angle
D14	Output text to screen
D15	Output text or parameter contents through the data interface
D19	Transfer numerical values or Q parameters to the PLC

Addresses

%	Program beginning
A	Swiveling axis around X
B	Swiveling axis around Y
C	Rotary axis around Z
D	Define Q-parameter functions
E	Tolerance for rounding arc with M112
F	Feed rate in mm/min in positioning blocks
F	Dwell time in seconds with G04
F	Scaling factor with G72
G	G functions (see list of G functions)
H	Polar coordinate angle
H	Angle of rotation with G73
I	X coordinate of the circle center or pole
J	Y coordinate of the circle center or pole
K	Z coordinate of the circle center or pole
L	Label number with G98
L	Jump to a label number
L	Tool length with G99
M	Miscellaneous function
N	Block number
P	Cycle parameter for fixed cycles
P	Value or Q parameter with Q parameter definitions
Q	Variable Q parameter
R	Polar coordinate radius with G10/G11/G12/ G13/G15/G16/
R	Circle radius with G02/G03/G05
R	Corner radius with G25/G26/G27
R	Chamfer length with G24
R	Tool radius with G99
S	Spindle speed in rpm
S	Angle for spindle orientation with G36
T	Tool number with G99
T	Tool call
T	Call next tool with G51
U	Parallel axis to X
V	Parallel axis to Y
W	Parallel axis to Z
X	X axis
Y	Y axis
Z	Z axis
*****	Character for end of block

Miscellaneous Functions M

M00	Stop program run/Stop spindle/Coolant off
M01	Optional program stop
M02	Stop program run/Stop spindle/Coolant off Jump back to block 1/Clear status display (depending on machine parameters)
M03	Spindle on clockwise
M04	Spindle on counterclockwise
M05	Optional stop
M06	Tool change/Stop program run (depending on machine parameters) Stop spindle
M08	Coolant on
M09	Coolant off
M13	Spindle on clockwise/Coolant on
M14	Spindle on counterclockwise/Coolant on
M30	Same function as M02
M89	Vacant miscellaneous function or cycle call, modally effective (depending on machine parameters)
M90	Constant contour speed at corners (effective only in lag mode)
M91	Within the positioning block: Coordinates are referenced to the machine datum
M92	Within the positioning block: The coordinates are referenced to a position defined by the machine tool builder
M93	Reserved
M94	Reduce rotary axis display to a value below 360°
M95	Reserved
M96	Reserved
M97	Machine small contour steps
M98	Suspend tool path compensation
M99	Cycle call, effective blockwise
M101	Automatic tool change after tool lifetime expires
M102	Reset M101
M103	Reduce the feed rate during plunging to factor F
M104	Reactivate most recently defined datum
M105	Machine with second k_V factor
M106	Machine with first k_V factor
M107	See User's Manual
M108	Reset M107
M109	Constant contouring speed of tool cutting edge on arcs (increasing and decreasing the feed rate)
M110	Constant contouring speed of tool cutting edge on arcs (only decreasing the feed rate)
M111	Reset M109/M110
M114	Automatic compensation of machine geometry when working with tilting axes

M115	Reset M114
M116	Feed rate for rotary axes in mm/min
M117	Reset M116
M118[1)]	Superimpose handwheel positioning during program run
M120[1)]	LOOK AHEAD: Calculate the radius-compensated tool path ahead of time
M124	Do not include points when executing non-compensated line blocks
M126	Permit zero crossover on 360° rotary axes
M127	Reset M126
M128	Retain position of tool tip when positioning tilting axes (TCPM)[2)]
M129	Reset M128
M130[1)]	Within the positioning block: points are referenced to the non-tilted coordinate system
M134	Exact stop when positioning with rotary axes
M135	Reset M134

M136	Feed rate F in microns per spindle revolution
M137	Feed rate F in millimeters per minute
M138	Selection of tilted axes for M114, M128 and the tilted working plane cycle
M140	Retraction from the contour in the positive tool axis direction
M141	Suppress touch probe monitoring
M142	Delete modal program information
M143	Delete basic rotation
M144	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block
M145	Reset M144
M200[1)]	Miscellaneous function for laser cutting machines
⋮	
M204[1)]	See User's Manual

[1)] Only with conversational programming
[2)] TCPM: Tool Center Point Management

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

Technical support FAX +49 (8669) 31-1000
E-Mail: service@heidenhain.de
Measuring systems ☏ +49 (8669) 31-3104
E-Mail: service.ms-support@heidenhain.de
TNC support ☏ +49 (8669) 31-3101
E-Mail: service.nc-support@heidenhain.de
NC programming ☏ +49 (8669) 31-3103
E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 (8669) 31-3102
E-Mail: service.plc@heidenhain.de
Lathe controls ☏ +49 (711) 952803-0
E-Mail: service.hsf@heidenhain.de

www.heidenhain.de

364 816-21 · SW05 · 7/2002 · pdf · Subject to change without notice